

US012089328B2

(12) **United States Patent**
Zenou et al.

(10) **Patent No.:** **US 12,089,328 B2**
(45) **Date of Patent:** **Sep. 10, 2024**

(54) **PCB PRODUCTION BY LASER SYSTEMS**

(71) Applicant: **IO Tech Group Ltd.**, Modiin (IL)

(72) Inventors: **Michael Zenou**, Hashmonaim (IL);
Guy Neshet, Nes Ziona (IL)

(73) Assignee: **IO Tech Group Ltd.**, Modiin (IL)

(*) Notice: Subject to any disclaimer, the term of this patent is extended or adjusted under 35 U.S.C. 154(b) by 0 days.

(21) Appl. No.: **18/499,009**

(22) Filed: **Oct. 31, 2023**

(65) **Prior Publication Data**

US 2024/0064897 A1 Feb. 22, 2024

Related U.S. Application Data

(62) Division of application No. 17/249,217, filed on Feb. 24, 2021, now Pat. No. 11,877,398.

(60) Provisional application No. 63/200,034, filed on Feb. 11, 2021.

(51) **Int. Cl.**

H05K 1/11 (2006.01)
H05K 3/34 (2006.01)
H05K 13/04 (2006.01)

(52) **U.S. Cl.**

CPC **H05K 1/111** (2013.01); **H05K 3/341** (2013.01); **H05K 13/046** (2013.01); **H05K 2201/10666** (2013.01)

(58) **Field of Classification Search**

CPC H05K 1/111
USPC 174/255
See application file for complete search history.

(56) **References Cited**

U.S. PATENT DOCUMENTS

4,098,612 A 7/1978 Rhodes et al.
4,174,973 A 11/1979 Rhodes et al.
5,059,266 A 10/1991 Yamane et al.
5,096,862 A 3/1992 Mathers et al.

(Continued)

FOREIGN PATENT DOCUMENTS

JP S5456847 A 5/1979
JP S62244650 A 10/1987

(Continued)

OTHER PUBLICATIONS

International Search Report and Written Opinion mailed Apr. 7, 2020, from the ISA/European Patent Office, for International Application No. PCT/IB2020/050664 (filed Jan. 28, 2020), 13 pages.

(Continued)

Primary Examiner — Stanley Tso

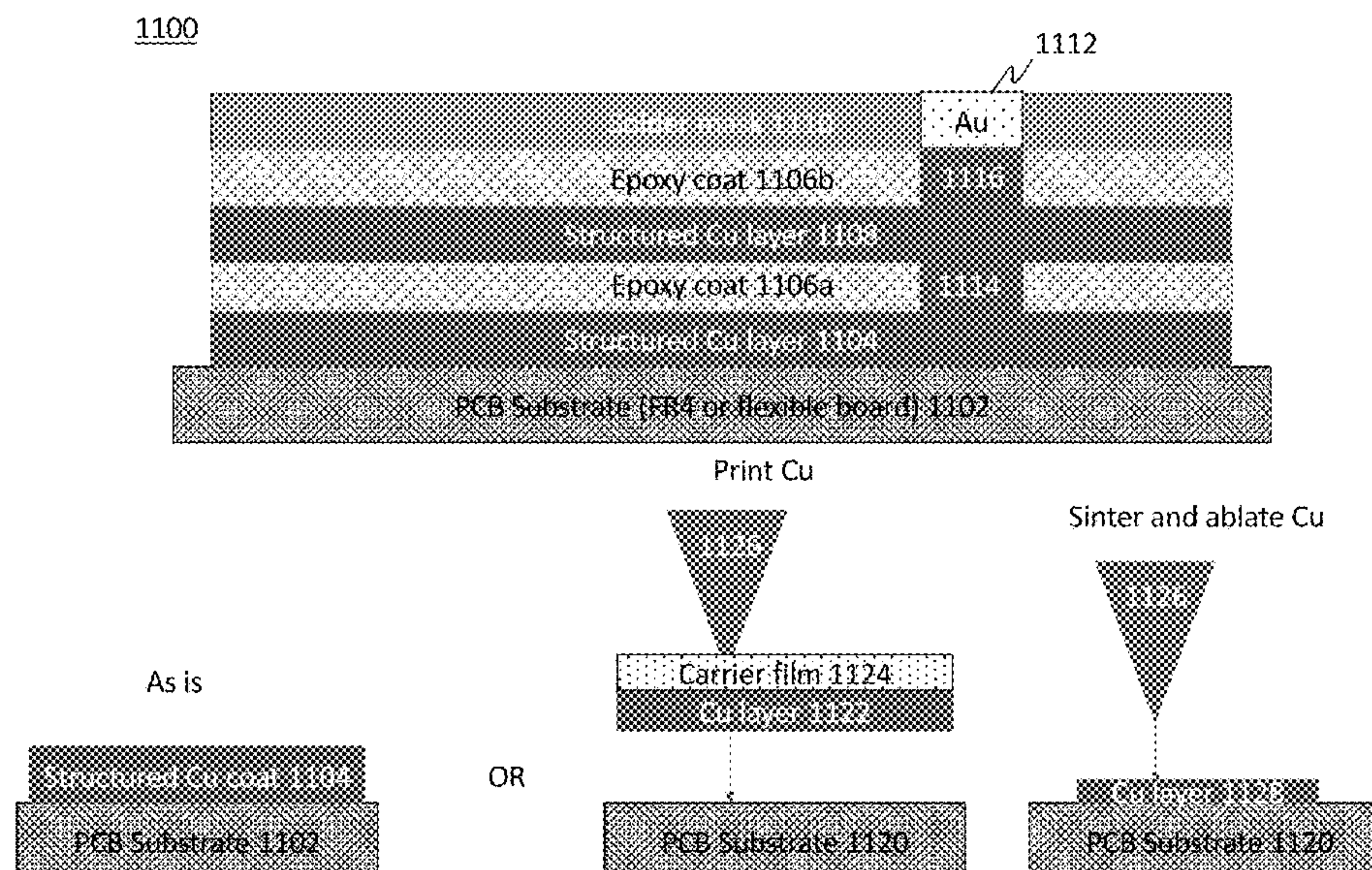
(74) *Attorney, Agent, or Firm* — Ascenda Law Group, PC

(57)

ABSTRACT

Systems and methods for printing a printed circuit board (PCB) from substrate to full integration utilize a laser-assisted deposition (LAD) system to print a flowable material on top of a substrate by laser jetting to create a PCB structure to be used as an electronic device. One such system for PCB printing includes a jet printing unit, an imaging unit, curing units, and a drilling unit to print metals and other materials (epoxies, solder masks, etc.) directly on a PCB substrate such as a glass-reinforced epoxy laminate material (e.g., FR4) or others. The jet printing unit can also be used for sintering and/or ablation of materials. Printed materials are cured by heating or by infrared (IR) or ultraviolet (UV) radiation. PCB s produced according to the present systems and methods may be single-sided or double-sided.

4 Claims, 52 Drawing Sheets



(56)

References Cited

U.S. PATENT DOCUMENTS

5,204,055 A 4/1993 Sachs et al.
 5,231,062 A 7/1993 Mathers et al.
 5,506,607 A 4/1996 Sanders, Jr. et al.
 5,590,454 A 1/1997 Richardson
 5,703,341 A 12/1997 Lowndes et al.
 5,740,051 A 4/1998 Sanders, Jr. et al.
 5,902,441 A 5/1999 Bredt et al.
 6,270,335 B2 8/2001 Leyden et al.
 6,322,728 B1 11/2001 Brodtkin et al.
 6,660,209 B2 12/2003 Leyden et al.
 6,921,500 B1 7/2005 Feenstra
 6,939,489 B2 9/2005 Moszner et al.
 6,955,776 B1 10/2005 Feenstra
 7,189,344 B2 3/2007 Rheinberger et al.
 7,198,879 B1 4/2007 Tredwell et al.
 9,592,105 B2 3/2017 Hauptmann et al.
 2002/0006532 A1 1/2002 Robin
 2002/0122898 A1 9/2002 Ringeisen et al.
 2004/0032459 A1 2/2004 Te
 2004/0094058 A1 5/2004 Kasperchik et al.
 2006/0051533 A1 3/2006 Song et al.
 2006/0072944 A1 4/2006 Sharma et al.
 2007/0046304 A1 3/2007 Mok et al.
 2008/0001297 A1 1/2008 Lotz et al.
 2008/0044602 A1 2/2008 Weed
 2011/0209749 A1* 9/2011 Yang H01L 31/022425
 361/679.01
 2012/0302003 A1* 11/2012 Shieh H01L 29/66969
 257/E21.46
 2014/0253392 A1 9/2014 Yarga et al.
 2014/0272121 A1 9/2014 Ng et al.
 2015/0033557 A1 2/2015 Kotler et al.
 2015/0037445 A1 2/2015 Murphy et al.
 2015/0201500 A1 7/2015 Shinar et al.
 2016/0198576 A1 7/2016 Lewis et al.
 2017/0189995 A1 7/2017 Zenou et al.
 2018/0024254 A1 1/2018 Roy et al.
 2018/0050550 A1 2/2018 Batt
 2018/0071989 A1 3/2018 Zenou et al.
 2019/0019736 A1* 1/2019 Schrauben H05K 3/046
 2019/0035550 A1 1/2019 Yosui
 2019/0312009 A1 10/2019 Arvin et al.
 2019/0322036 A1 10/2019 Zenou et al.
 2020/0261191 A1 8/2020 Zenou et al.

2020/0350275 A1 11/2020 Zenou et al.
 2021/0059045 A1 2/2021 Mikage et al.
 2022/0208701 A1 6/2022 Koduri

FOREIGN PATENT DOCUMENTS

WO 2009/035854 A1 3/2009
 WO 2014/078537 A1 5/2014
 WO 2020/205691 A1 10/2020

OTHER PUBLICATIONS

International Search Report and Written Opinion mailed Jul. 18, 2019, from the ISA/European Patent Office, for International Patent Application No. PCT/IB2019/053086 (filed Apr. 15, 2019), 15 pages.
 International Search Report and Written Opinion mailed May 13, 2022, from the ISA/European Patent Office, for International Patent Application No. PCT/IB2022/051215 (filed Feb. 10, 2022), 12 pgs.
 Written Opinion of the International Preliminary Examining Authority mailed Feb. 17, 2020, from the IPEA/European Patent Office, for International Patent Application No. PCT/IB2019/053086 (filed Apr. 15, 2019), 7 pages.
 International Preliminary Report on Patentability mailed Sep. 24, 2020, from the IPEA/European Patent Office, for International Patent Application No. PCT/IB2019/053086 (filed Apr. 15, 2019), 12 pages.
 International Preliminary Report on Patentability mailed Jan. 18, 2023, from the IPEA/European Patent Office, for International Patent Application No. PCT/IB2022/051215 (filed Feb. 10, 2022), 51 pgs.
 Amendment filed Jan. 6, 2023, for U.S. Appl. No. 17/649,100, filed Jan. 27, 2022, 5 pgs.
 Non-Final Office Action dated Oct. 6, 2022, for U.S. Appl. No. 17/649,100, filed Jan. 27, 2022, 9 pgs.
 Amendment filed Sep. 12, 2023, for U.S. Appl. No. 17/249,217, filed Feb. 24, 2021, 10 pgs.
 Non-Final Office Action dated Jun. 14, 2023, for U.S. Appl. No. 17/249,217, filed Feb. 24, 2021, 27 pgs.
 Notice of Allowance mailed Oct. 30, 2023, for U.S. Appl. No. 17/249,217, filed Feb. 24, 2021, 7 pgs.
 Amendment filed Feb. 26, 2024, for U.S. Appl. No. 17/649,100, filed Jan. 27, 2022, 5 pgs.
 Non-Final Office Action dated Dec. 7, 2023, for U.S. Appl. No. 17/649,100, filed Jan. 27, 2022, 12 pgs.

* cited by examiner

100a

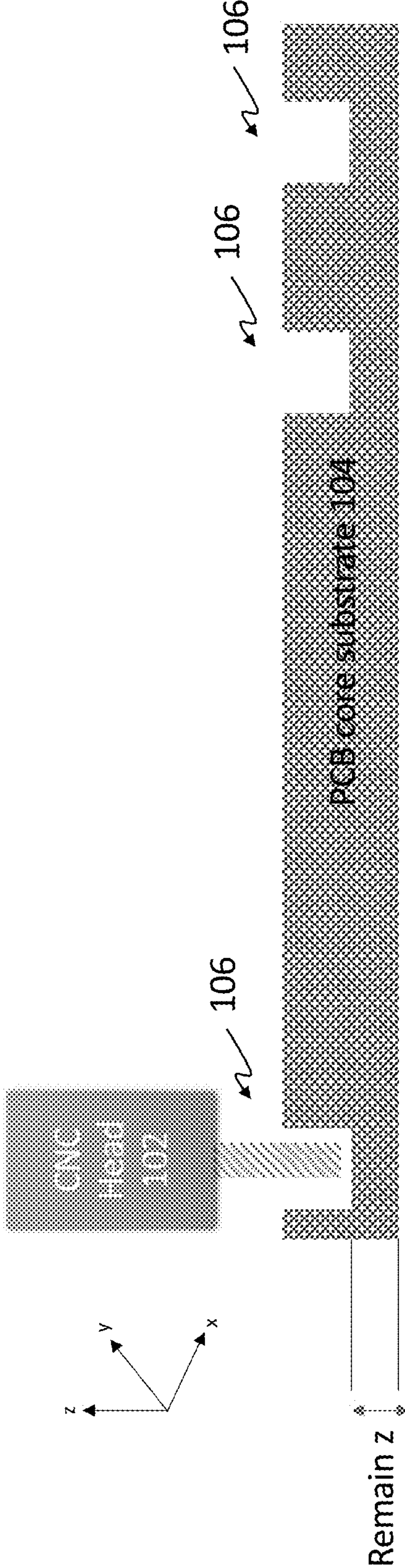


Figure 1a

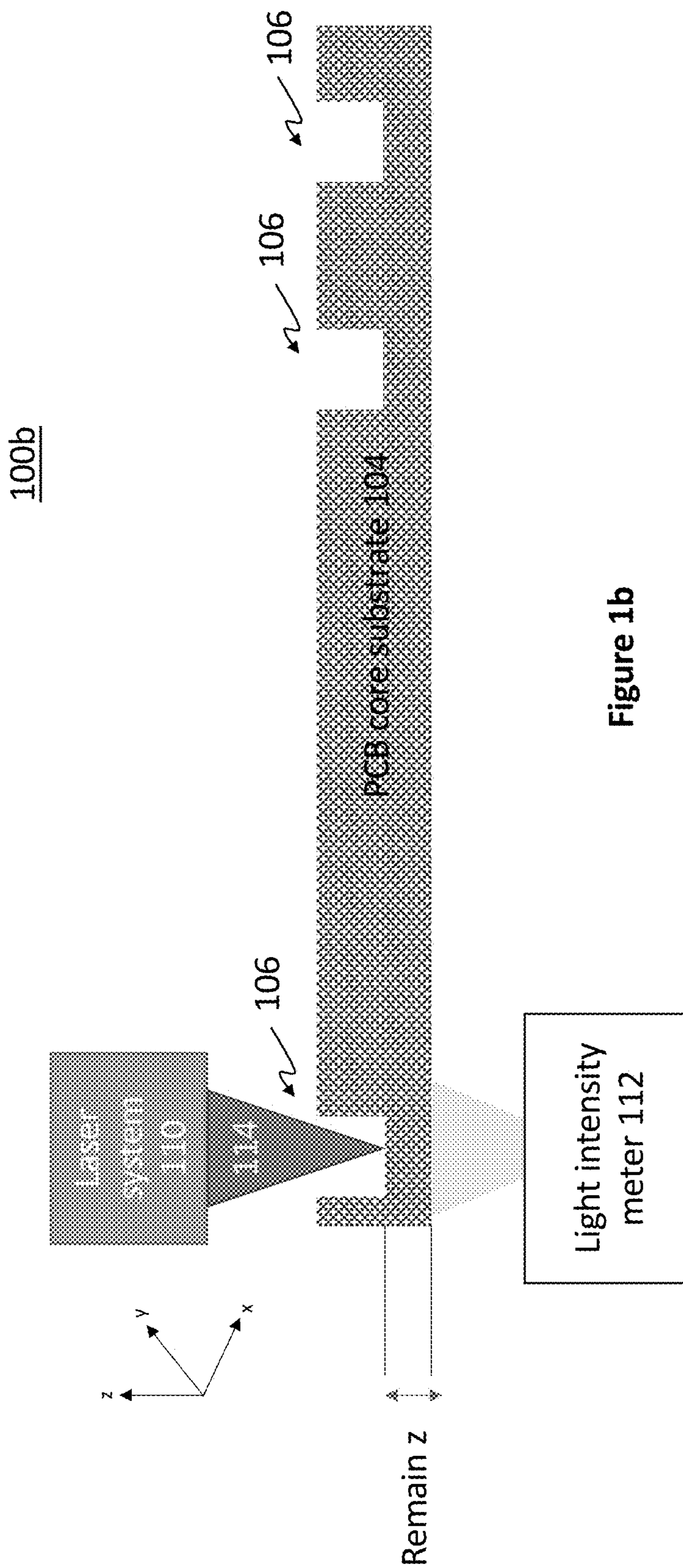


Figure 1b

100c

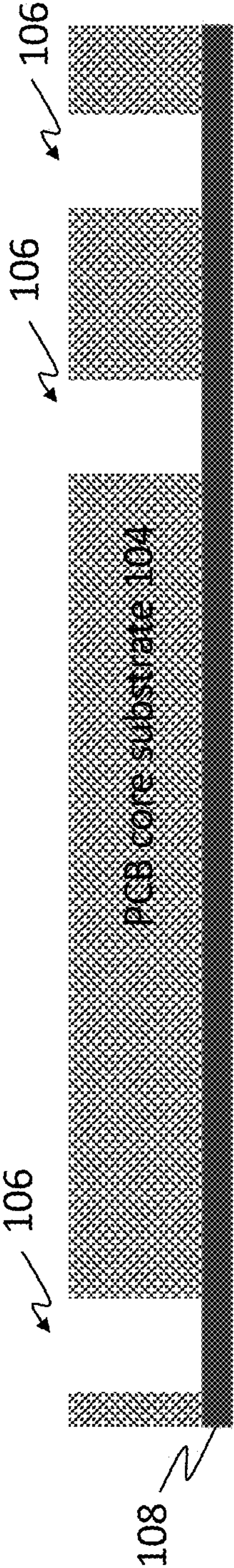


Figure 1c

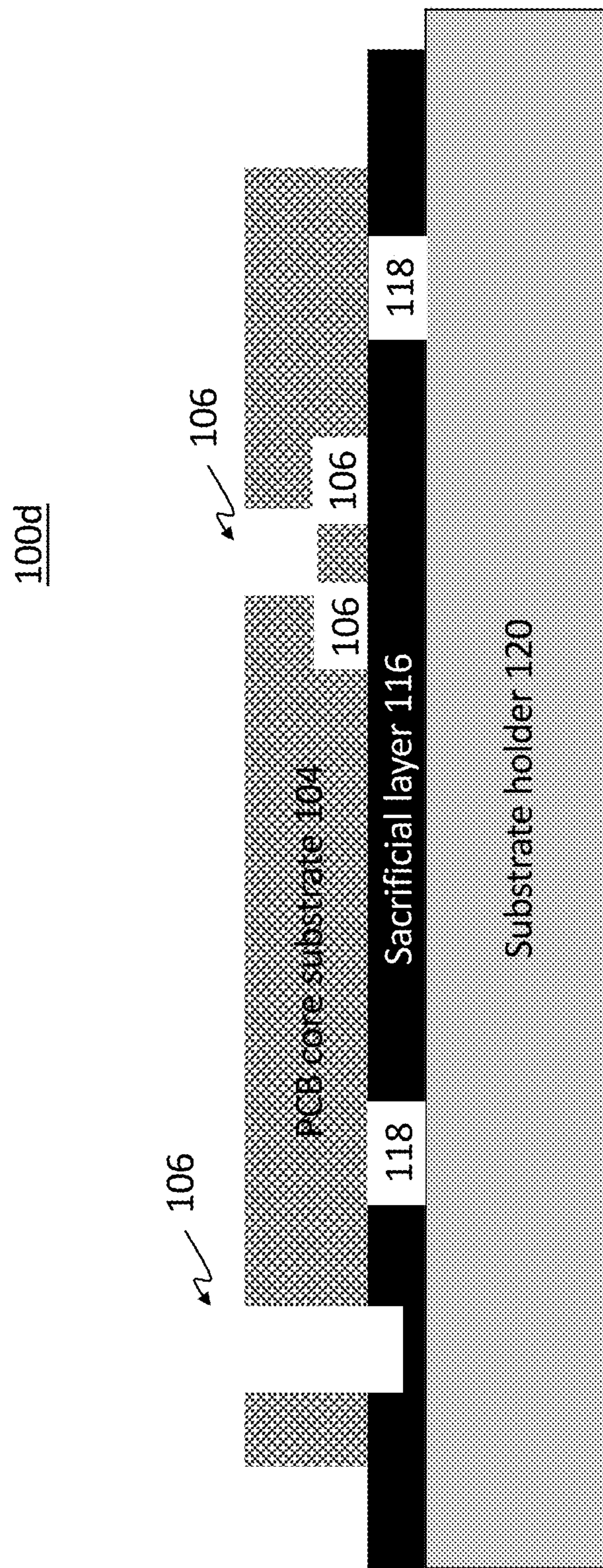


Figure 1d

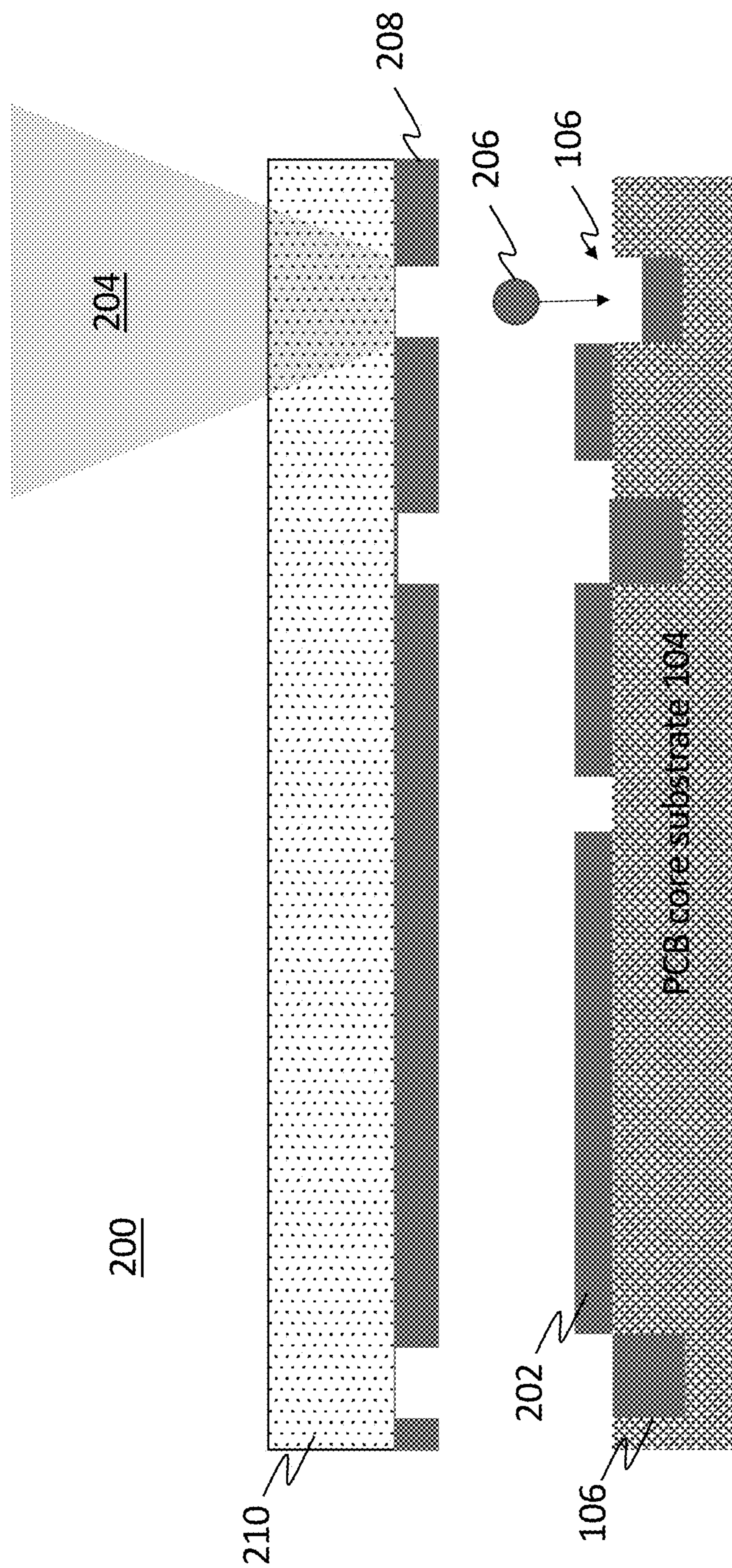


Figure 2a

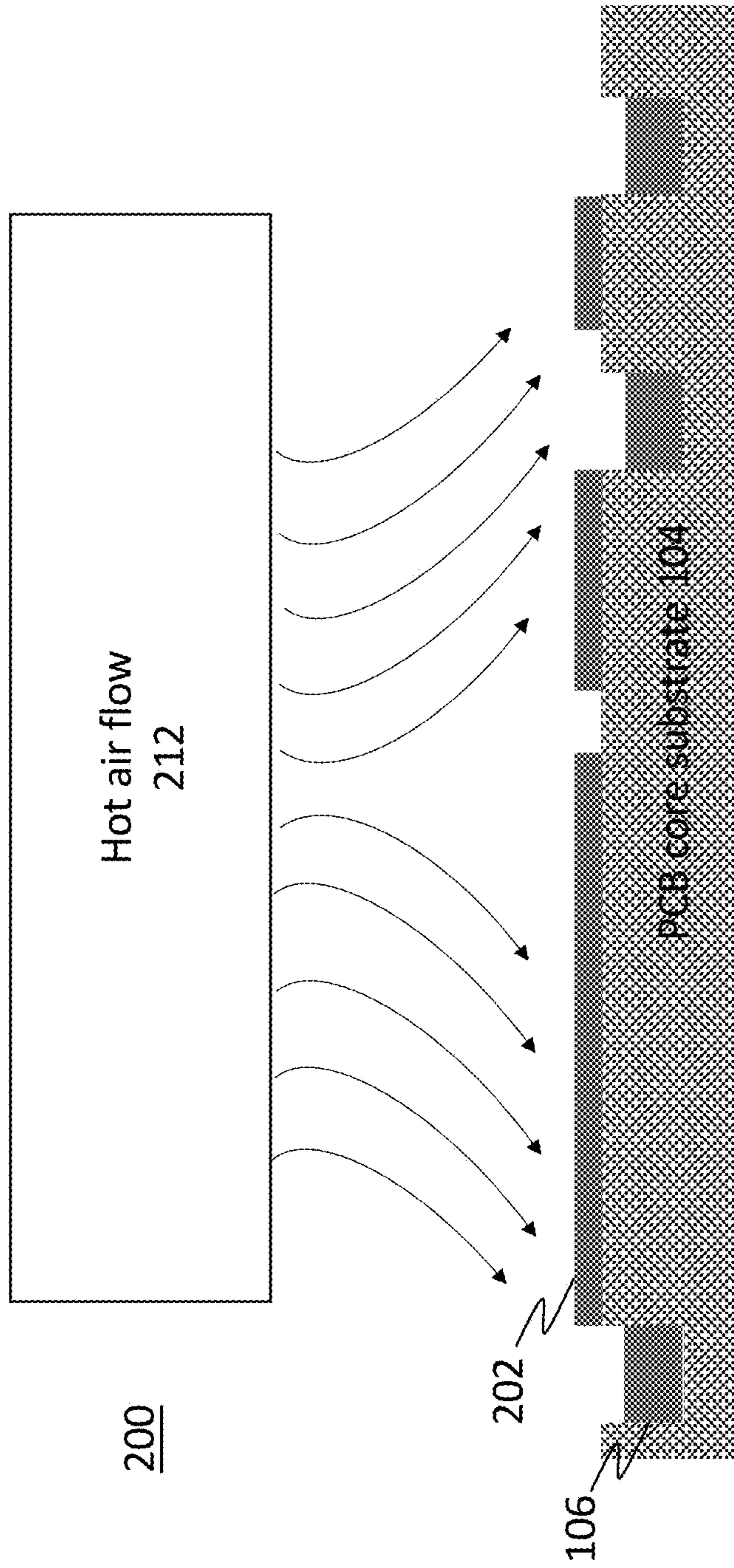


Figure 2b

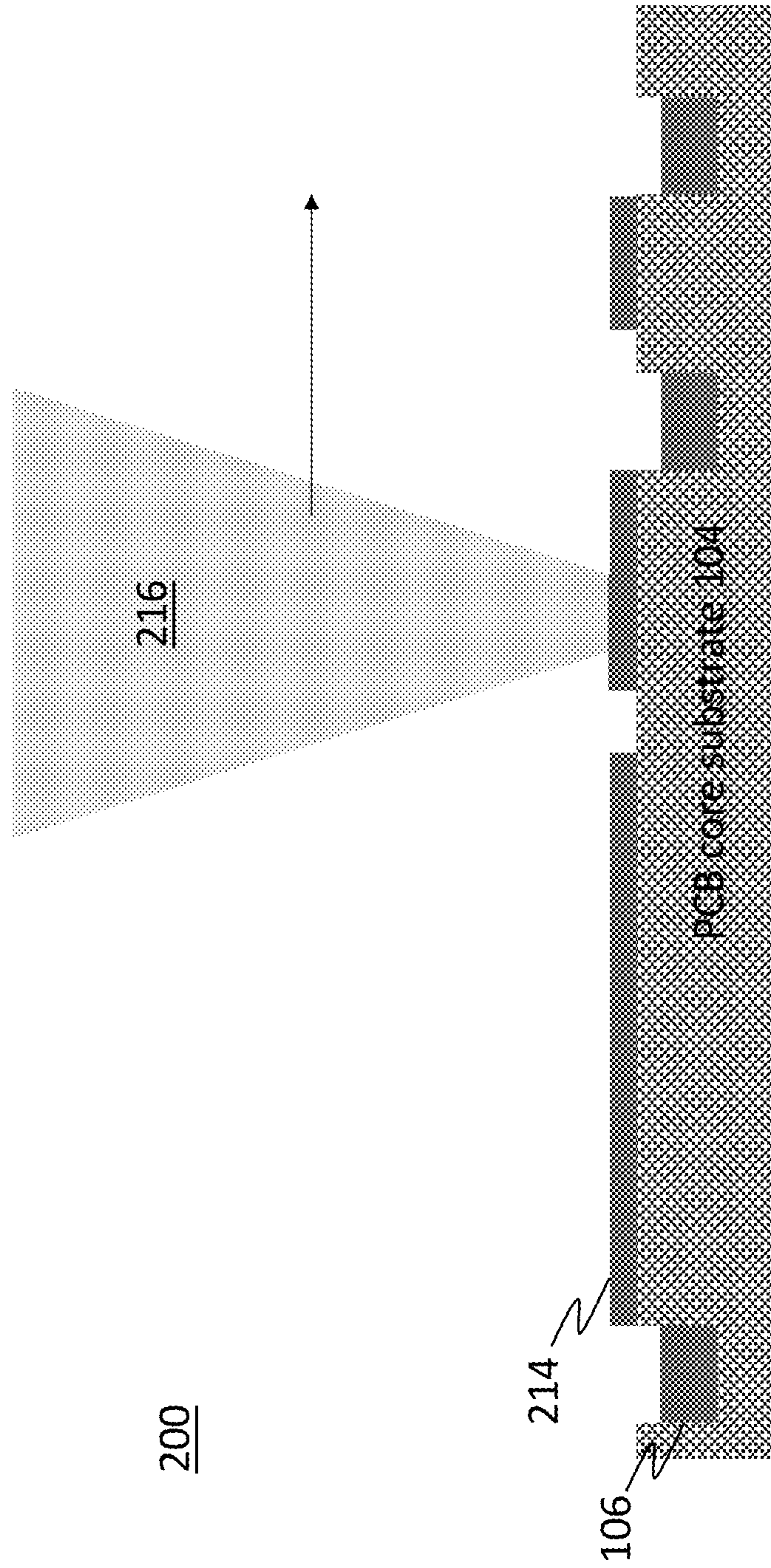


Figure 2c

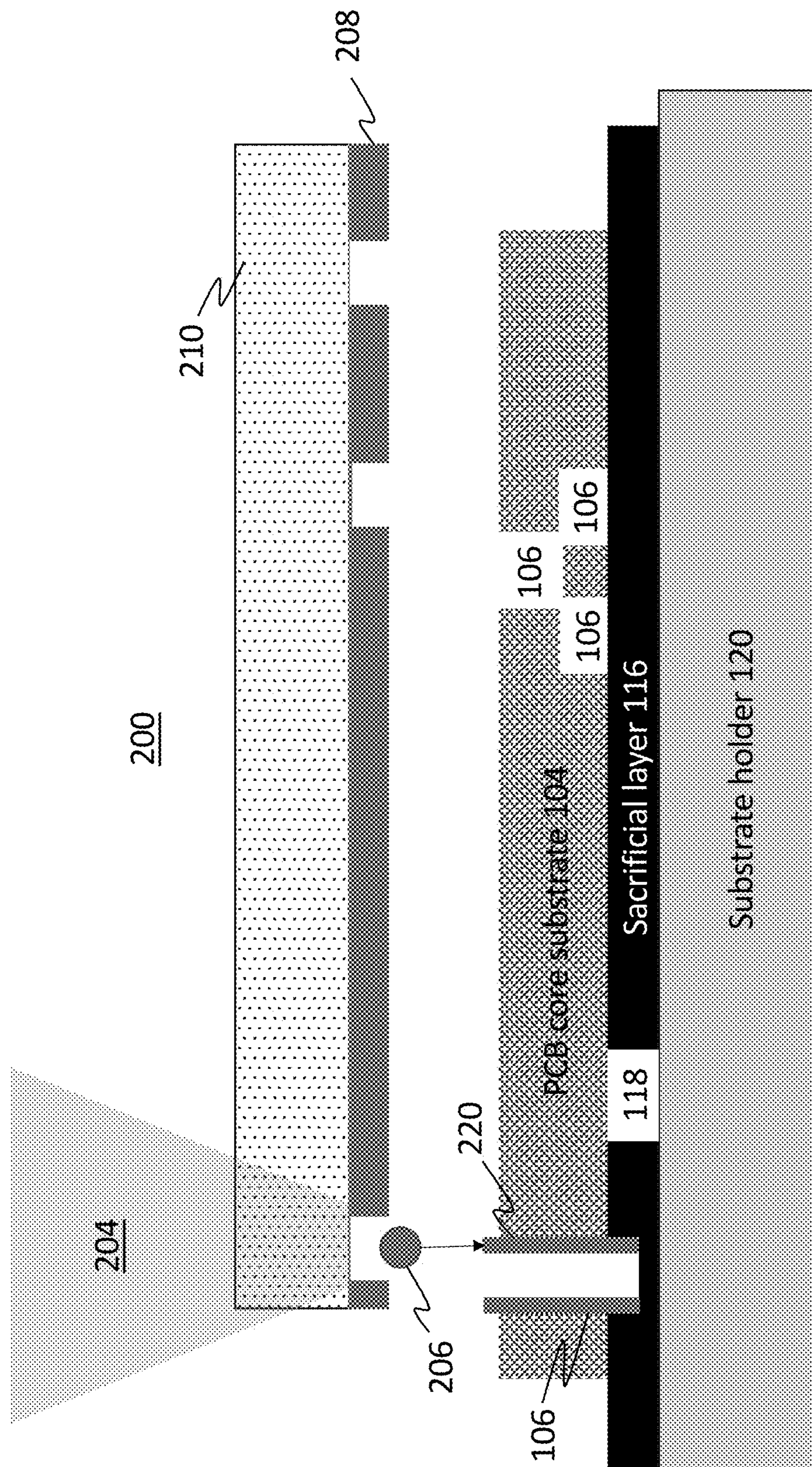


Figure 2d

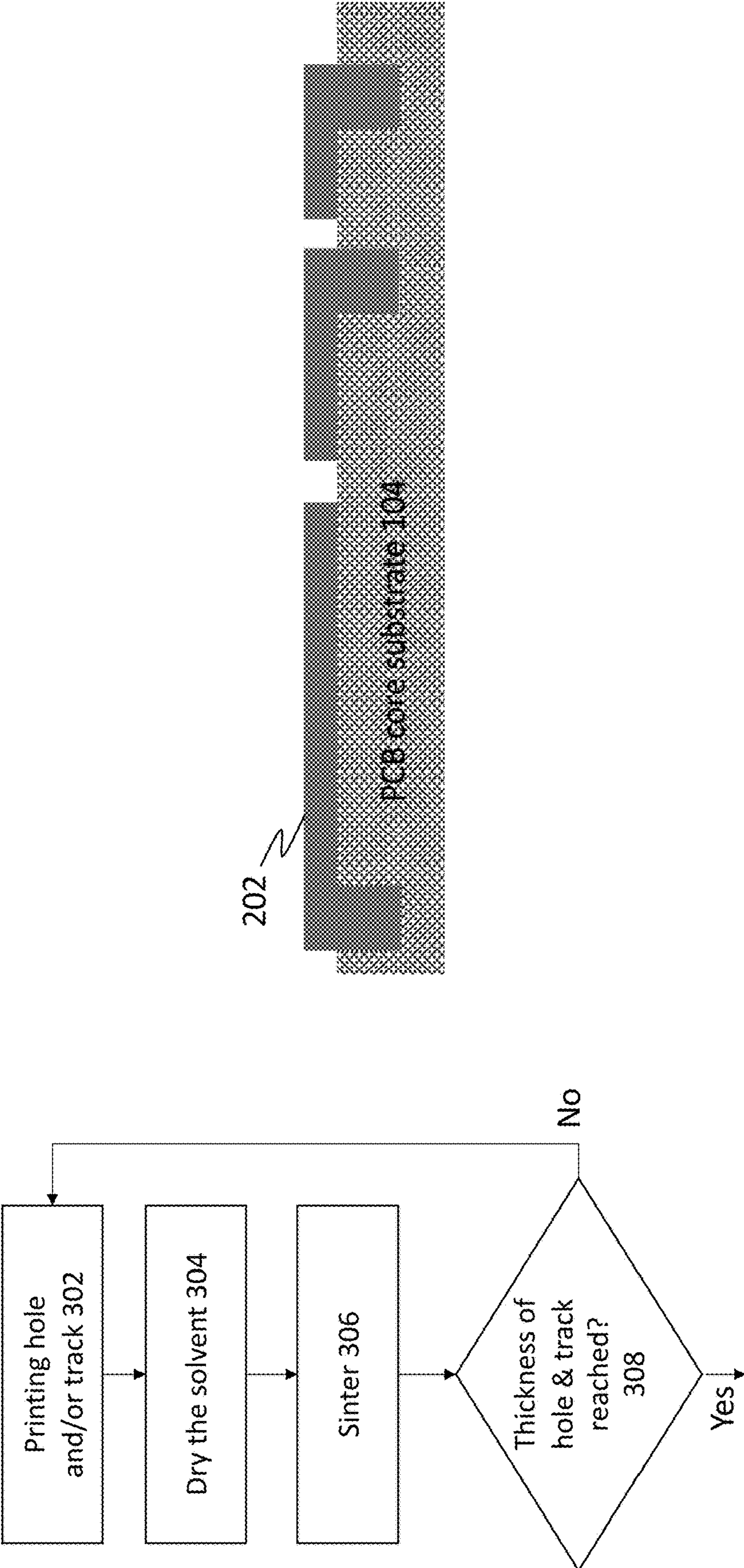


Figure 3a

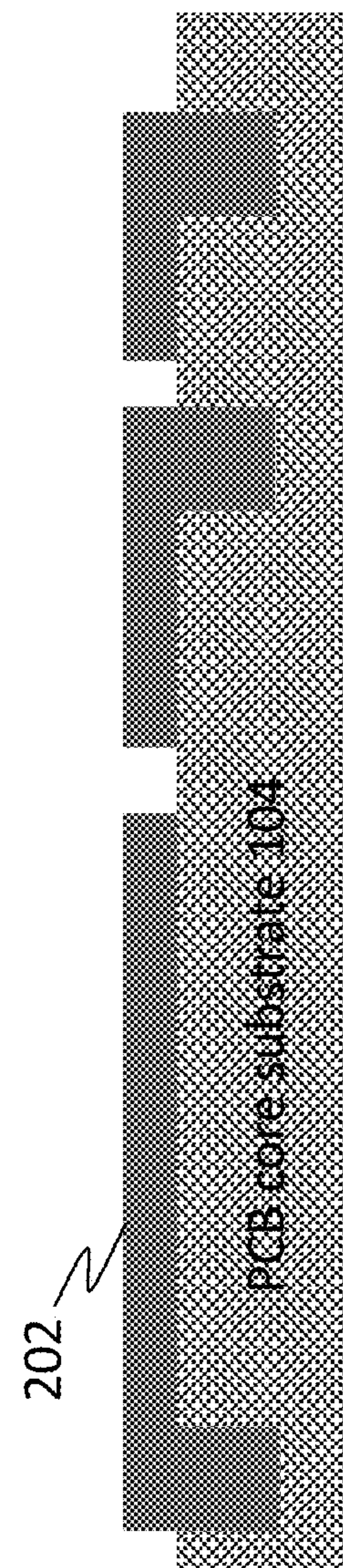
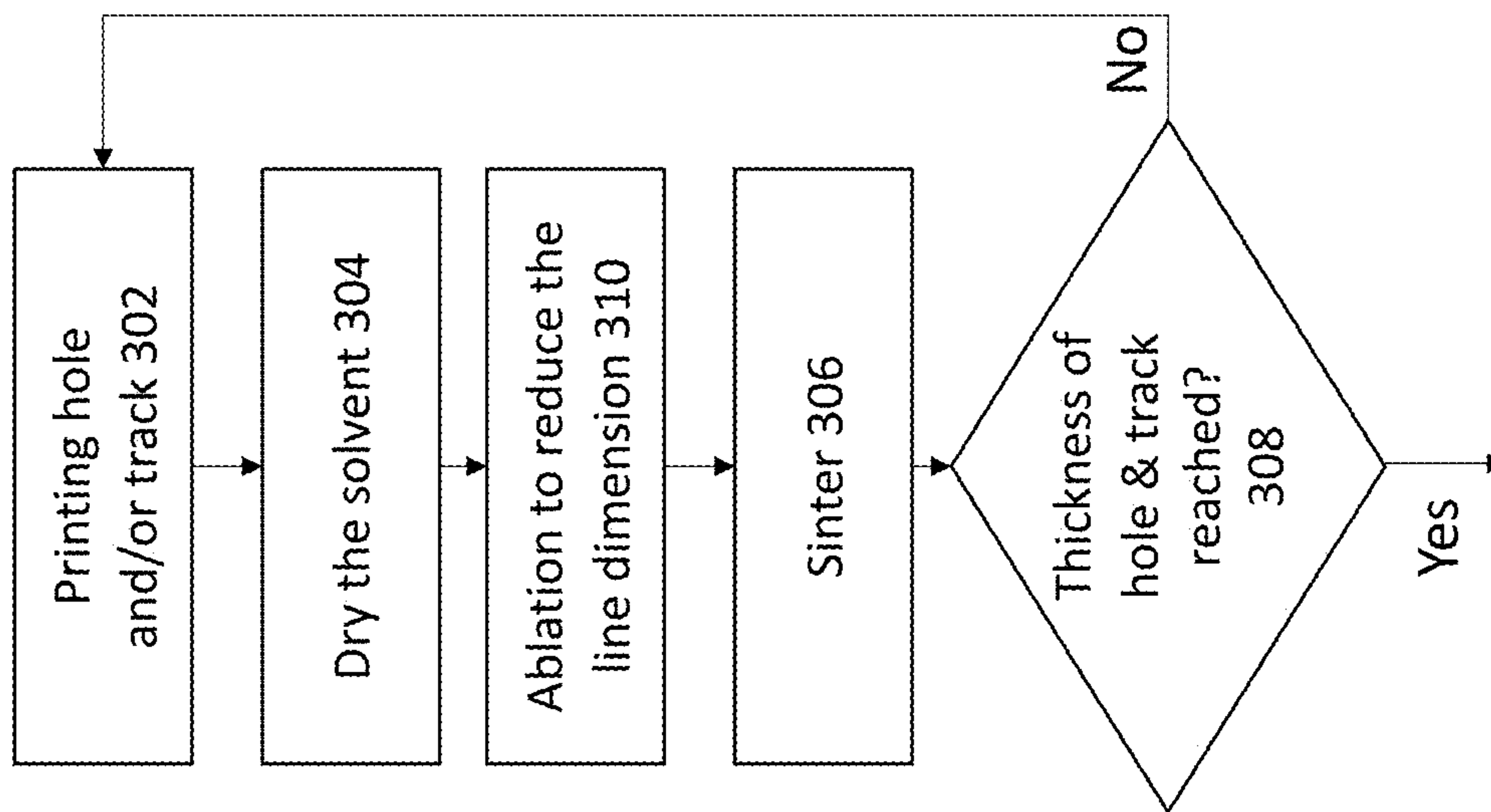


Figure 3b

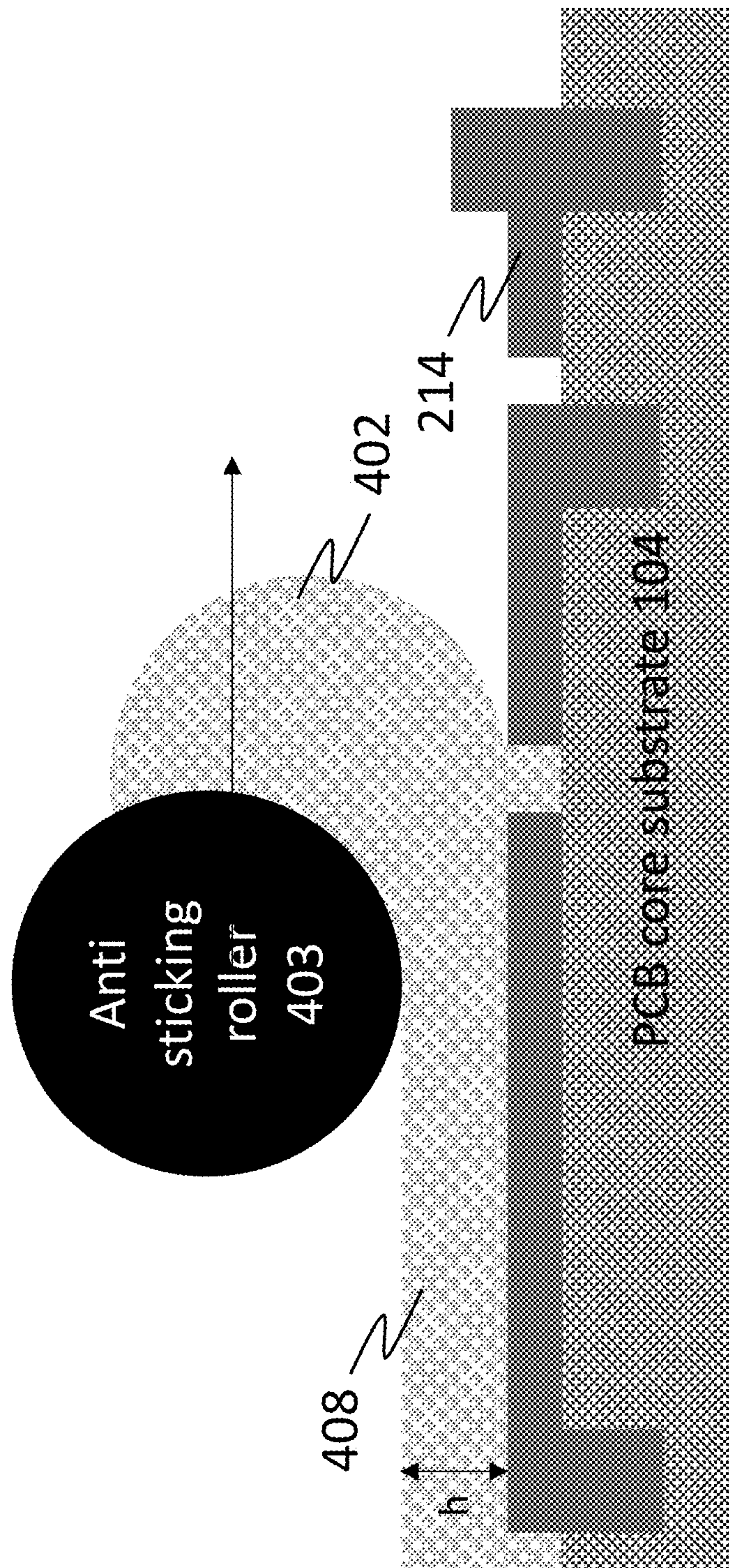


Figure 4a

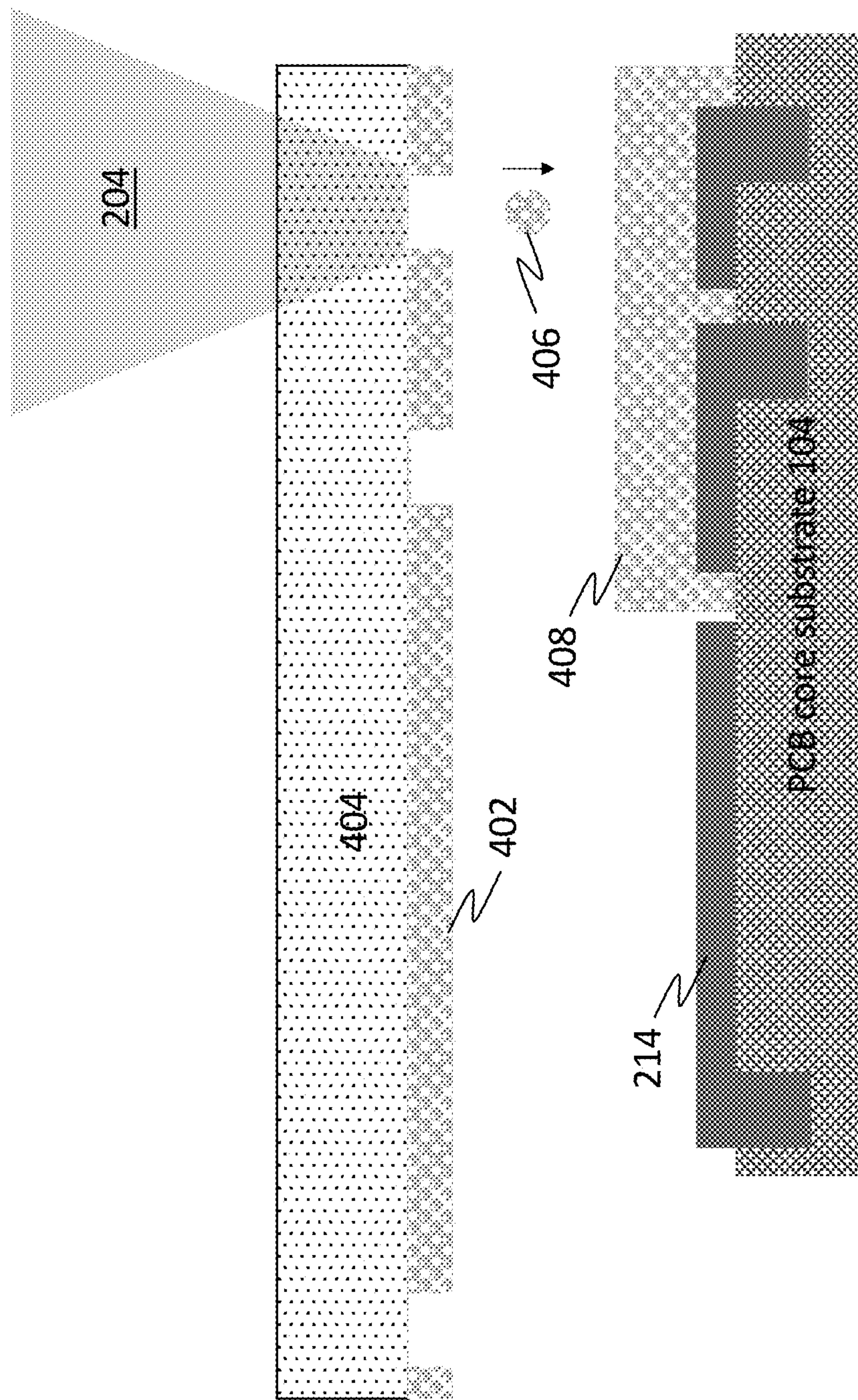


Figure 4b

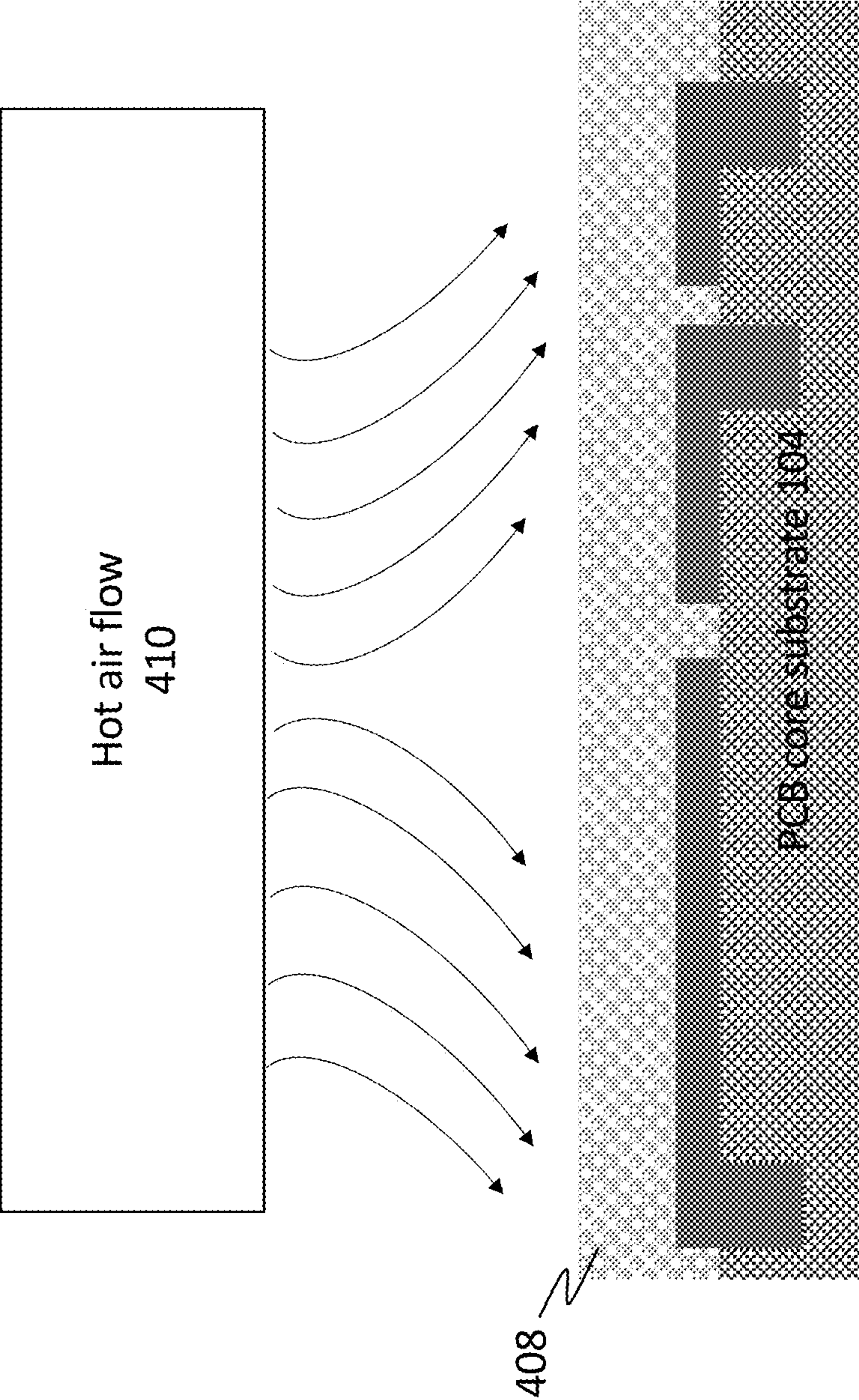


Figure 4c

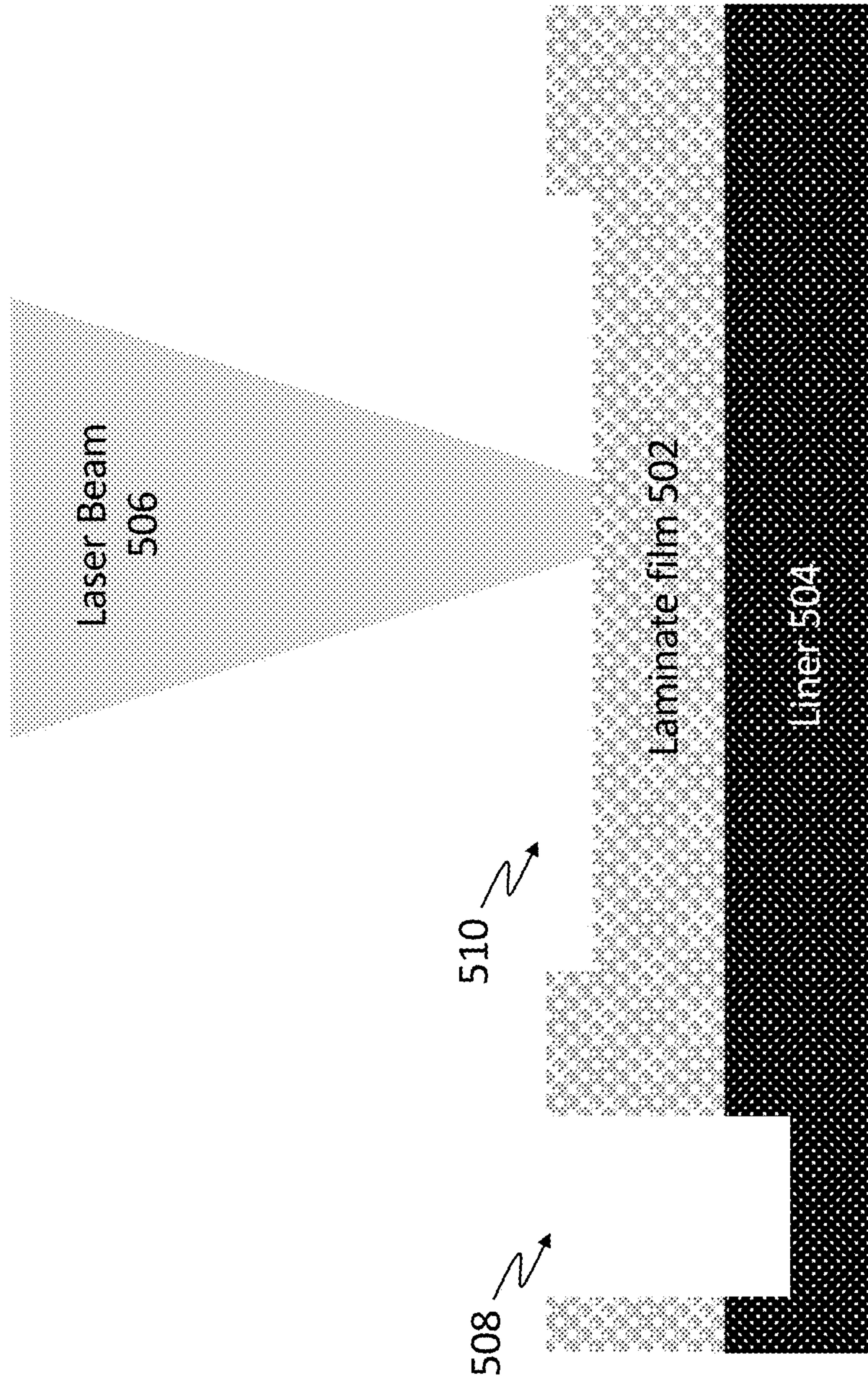


Figure 5a

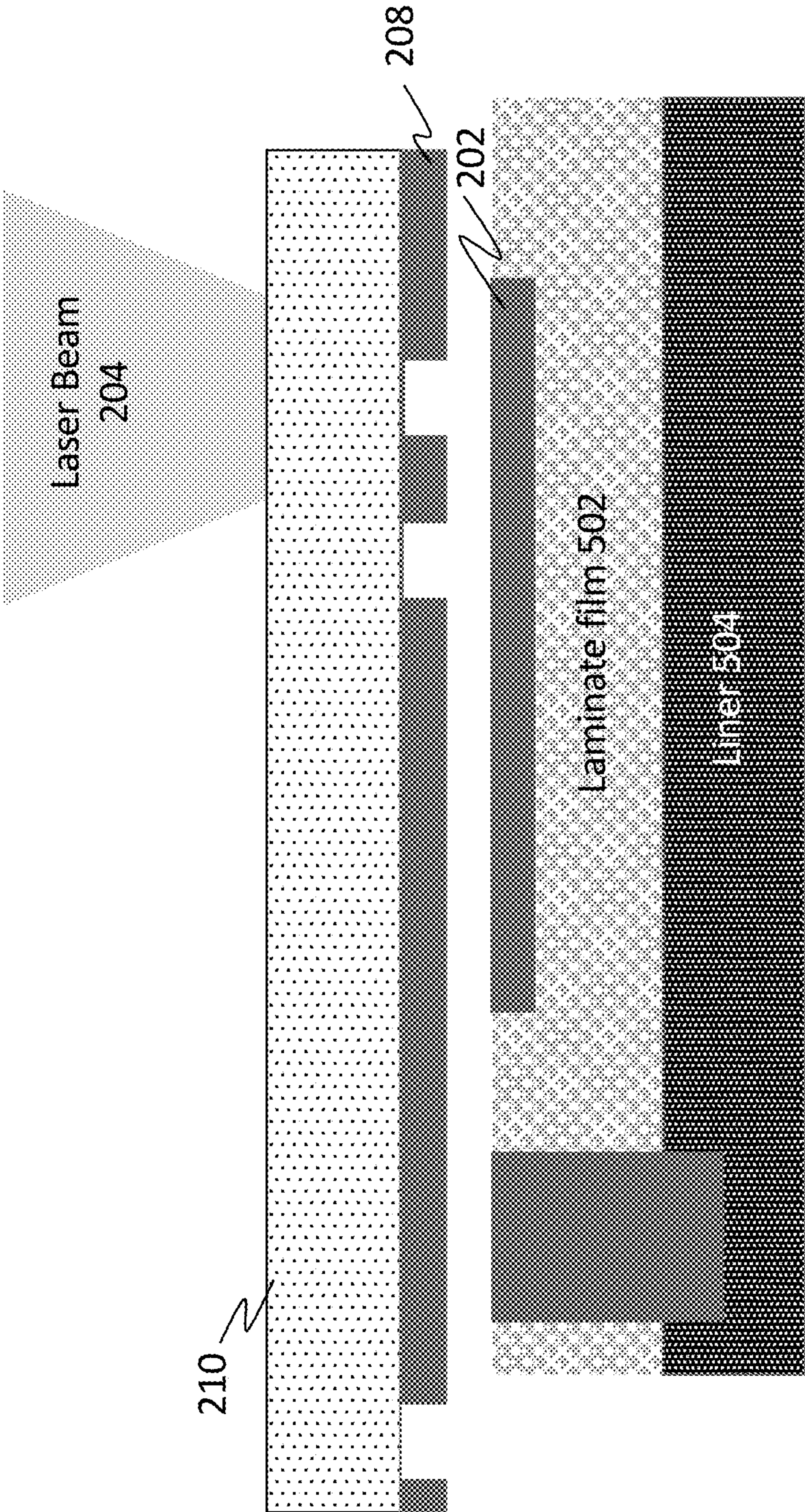


Figure 5b

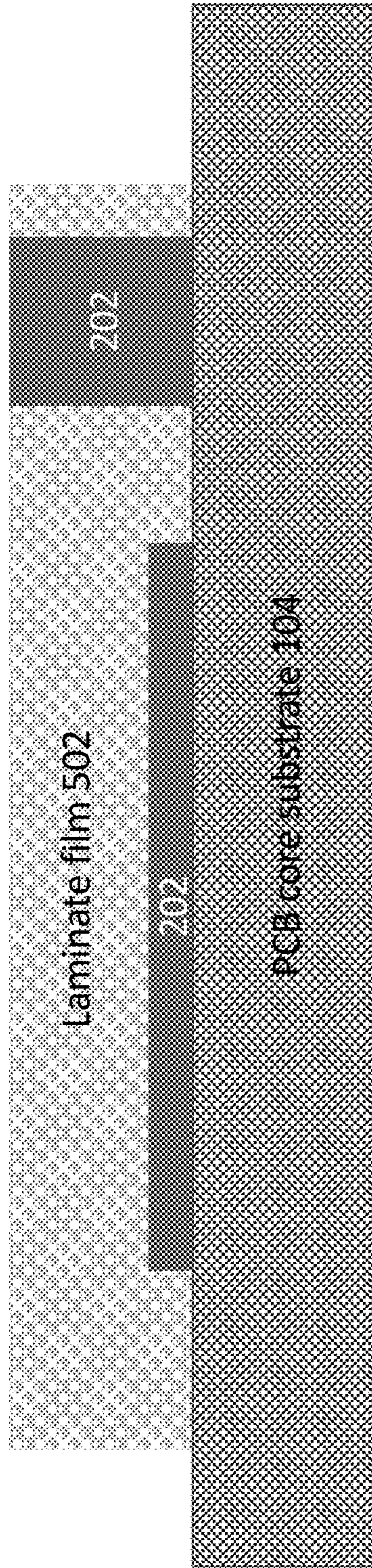


Figure 5c

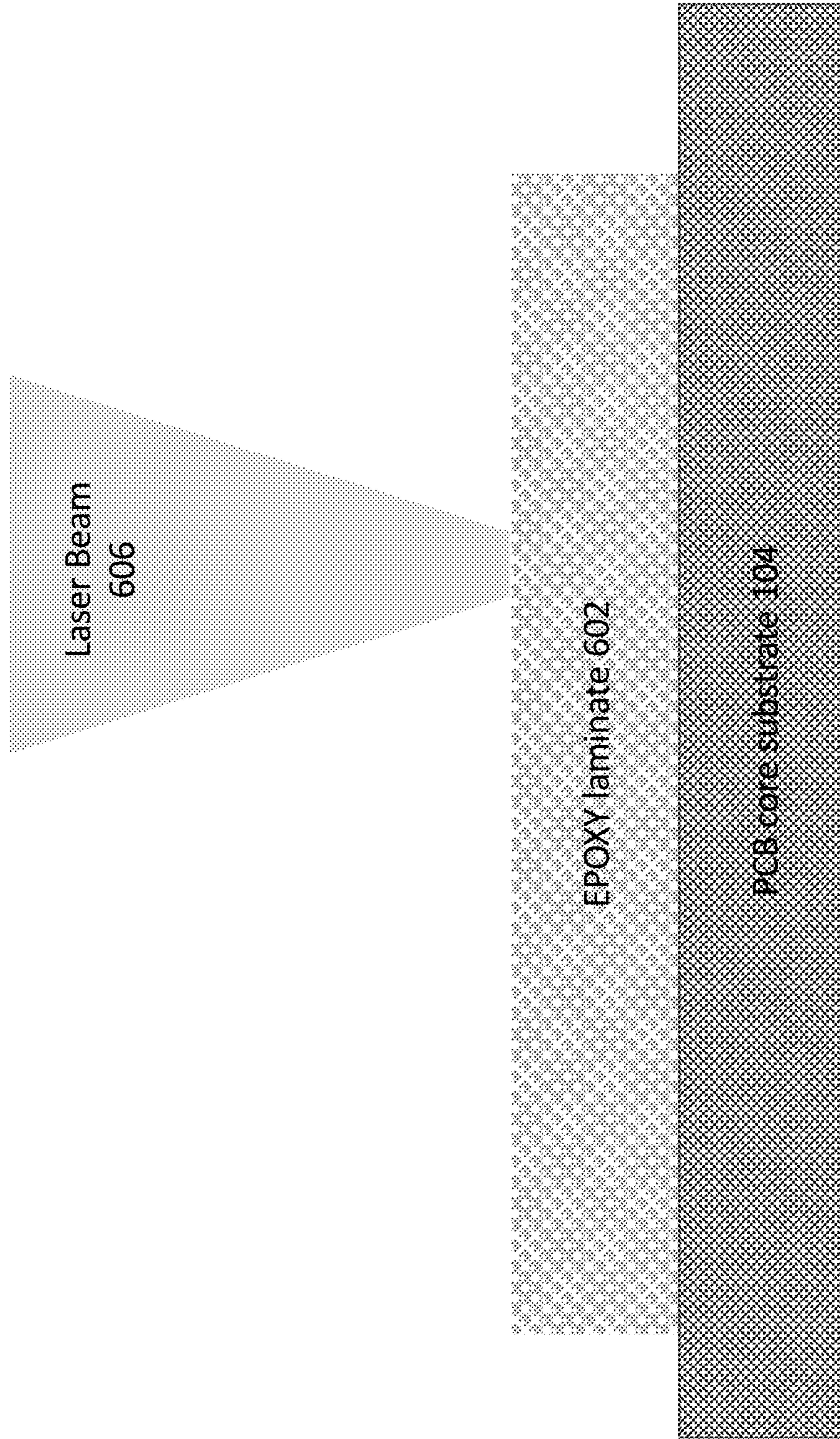


Figure 6a

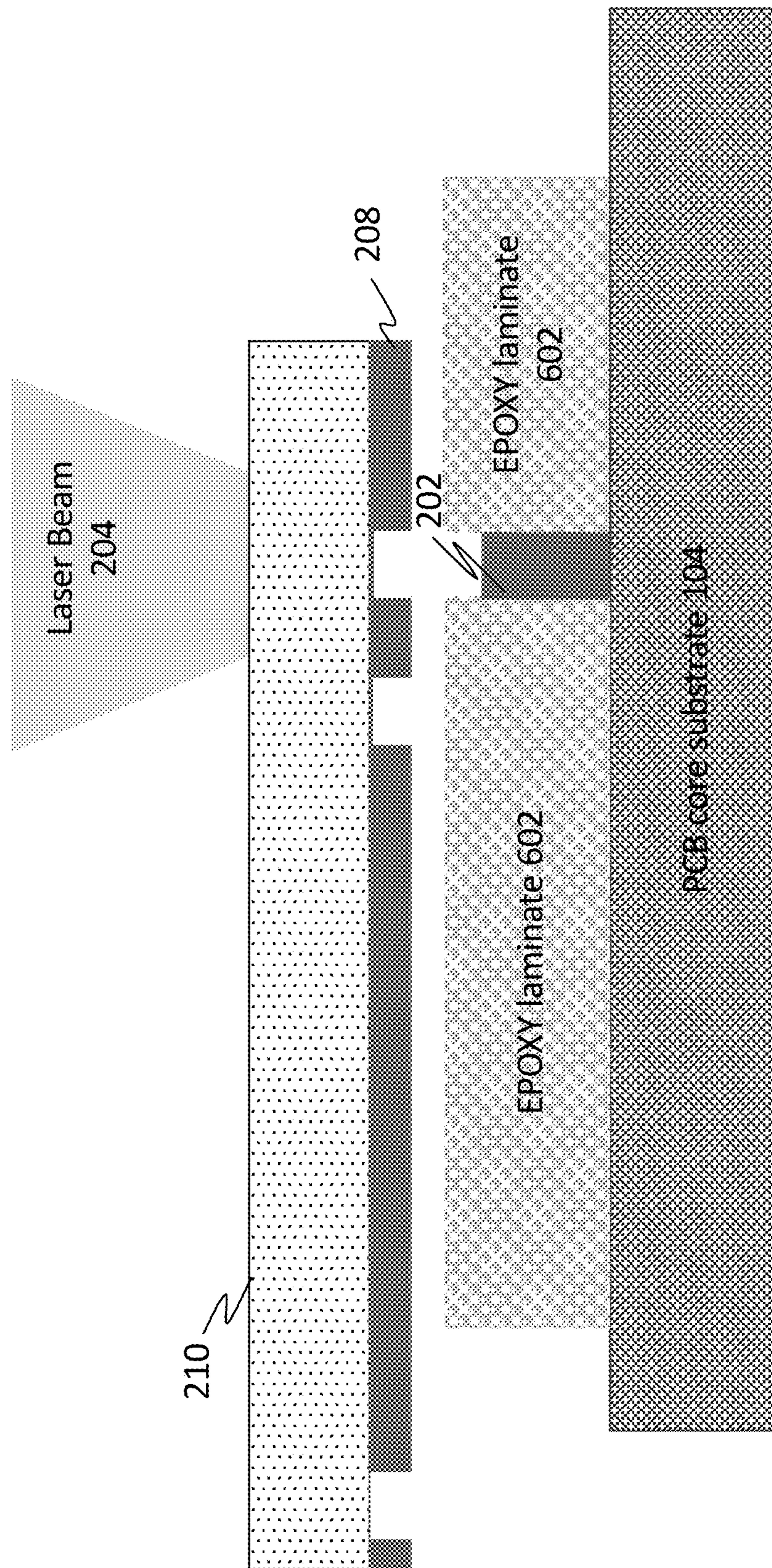


Figure 6b

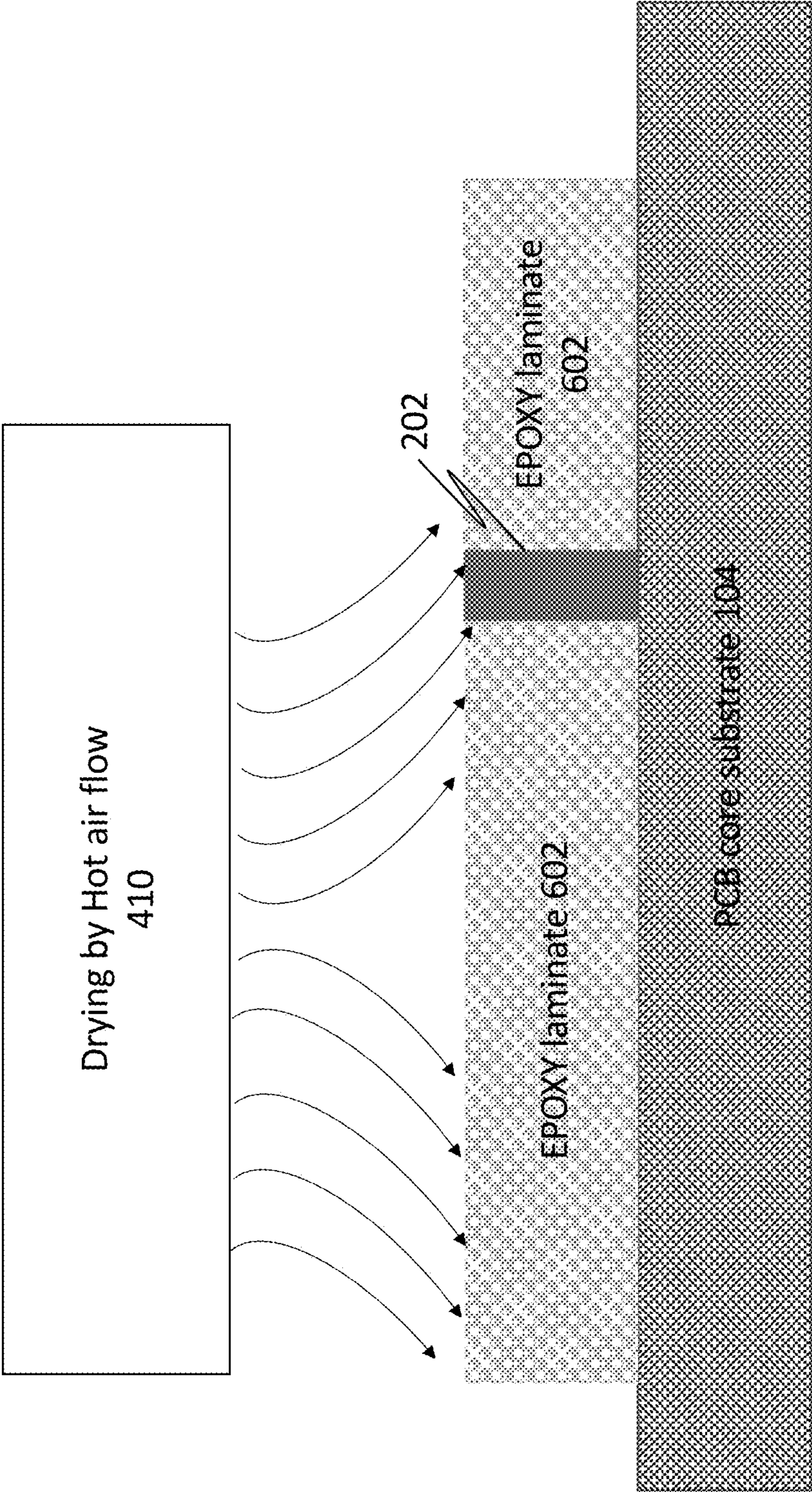


Figure 6c

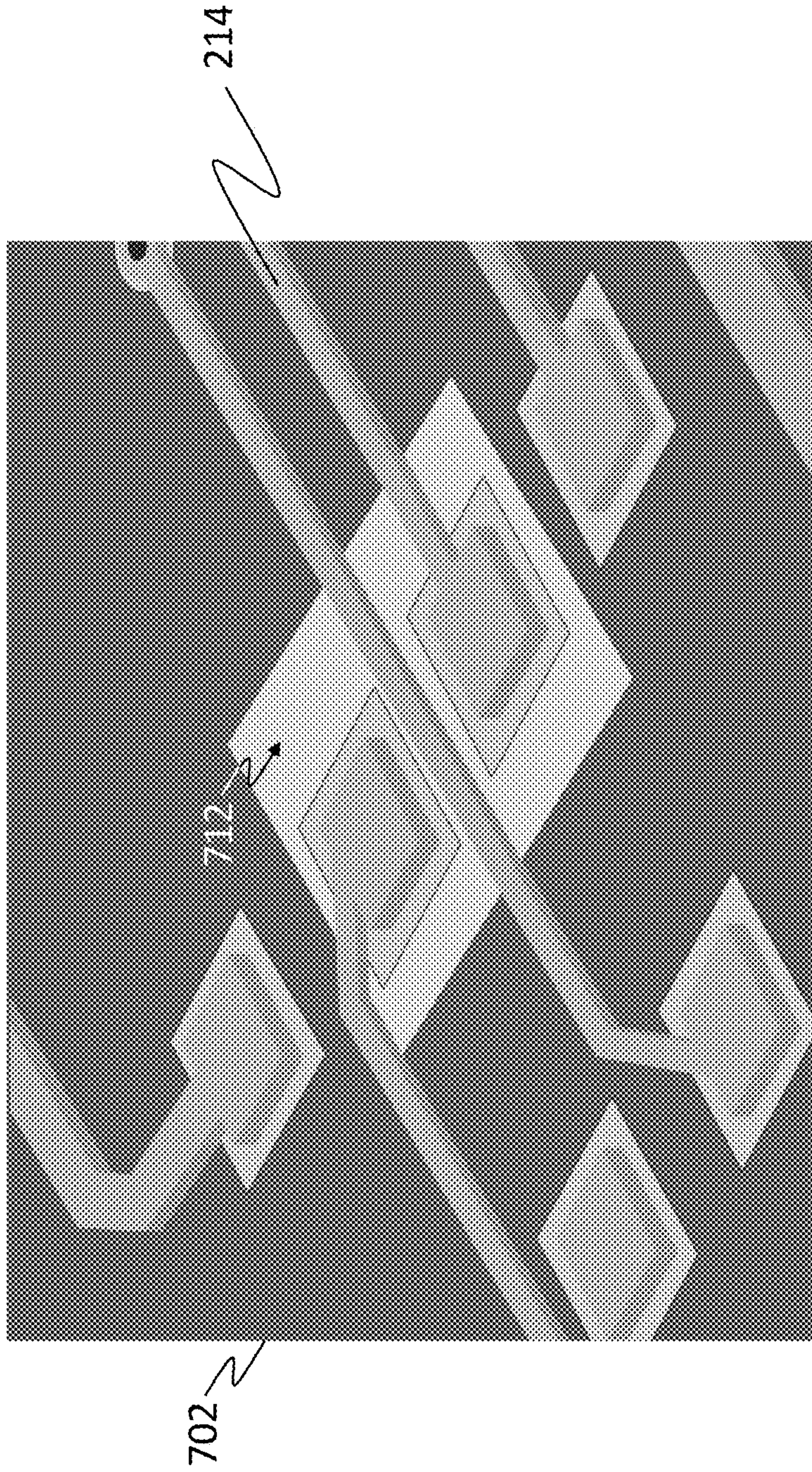


Figure 7b

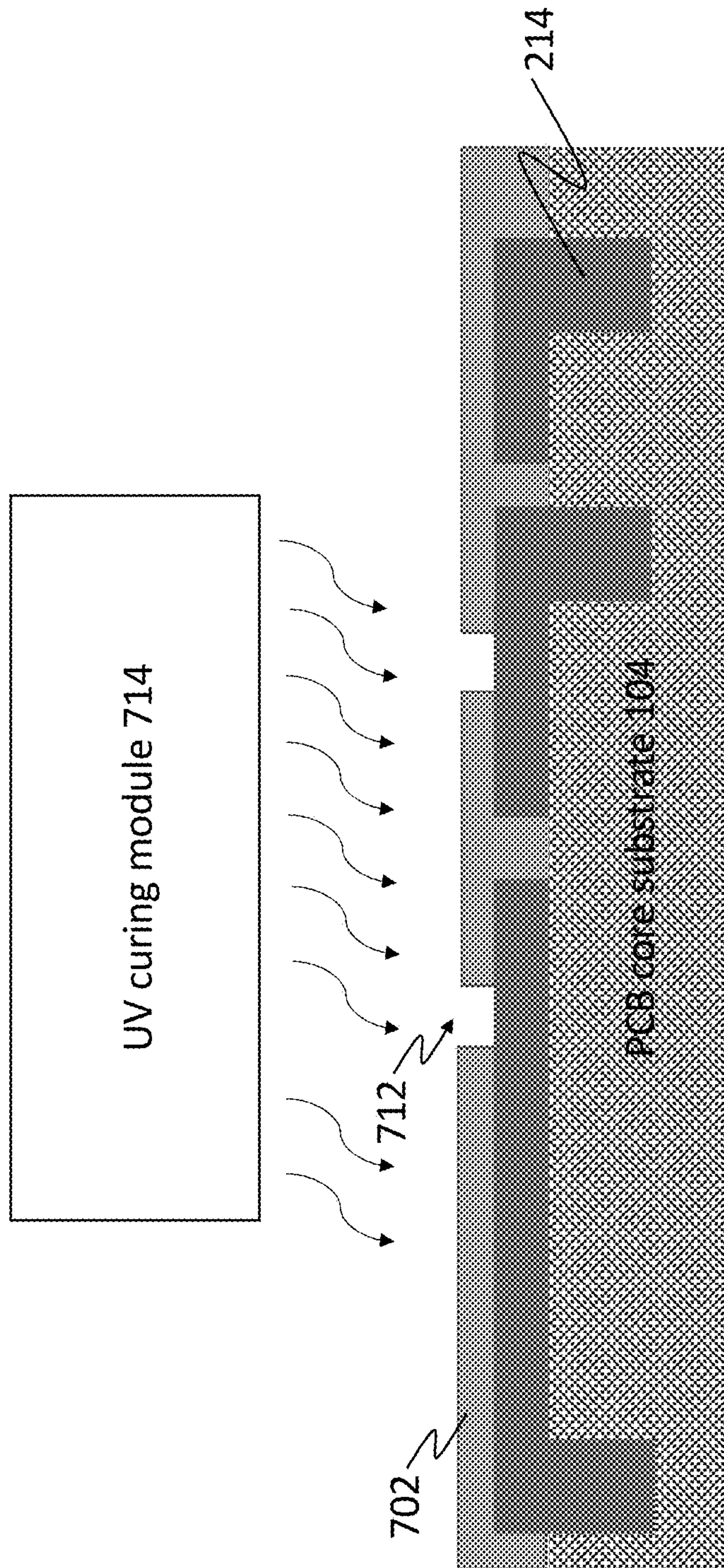


Figure 7c

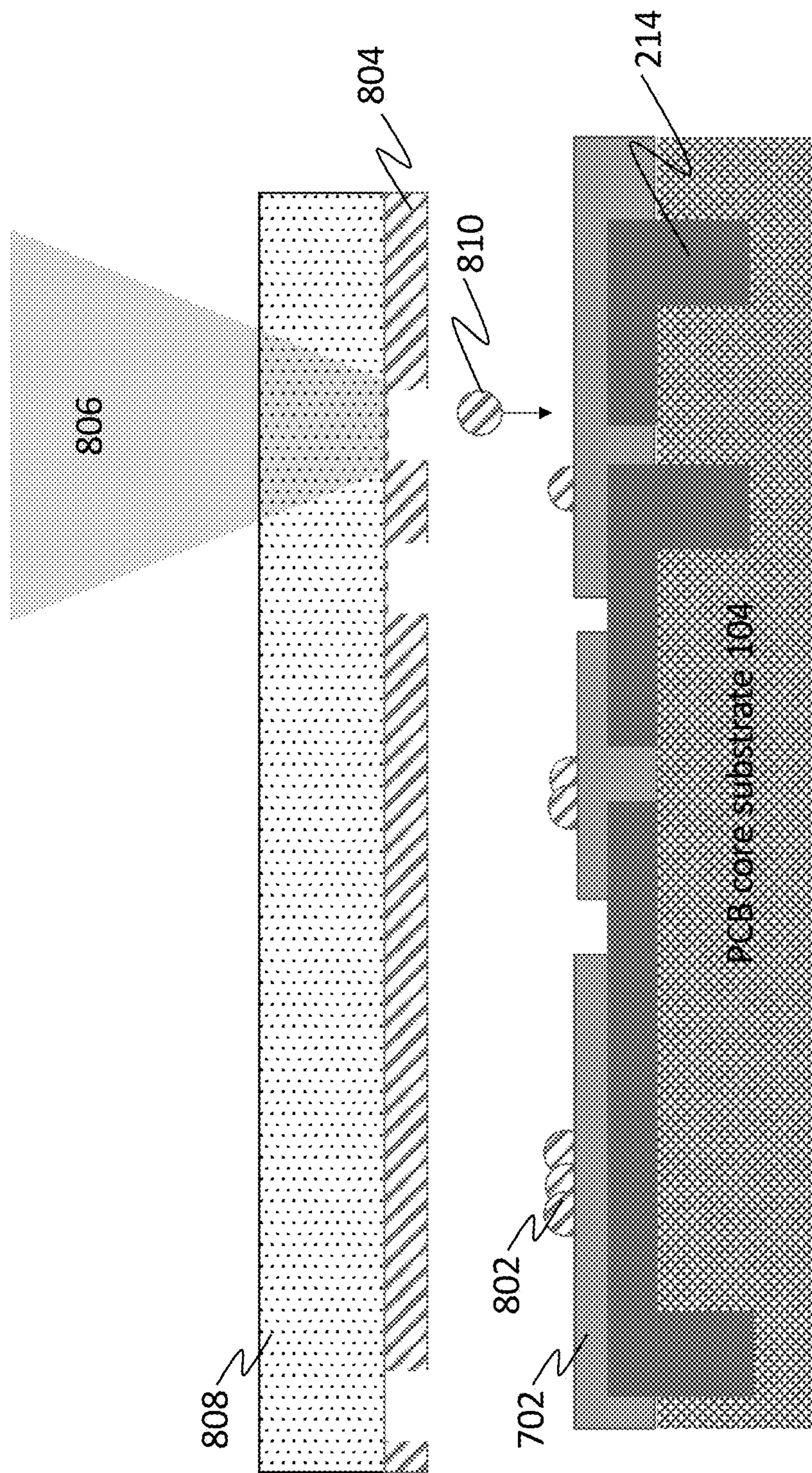


Figure 8a

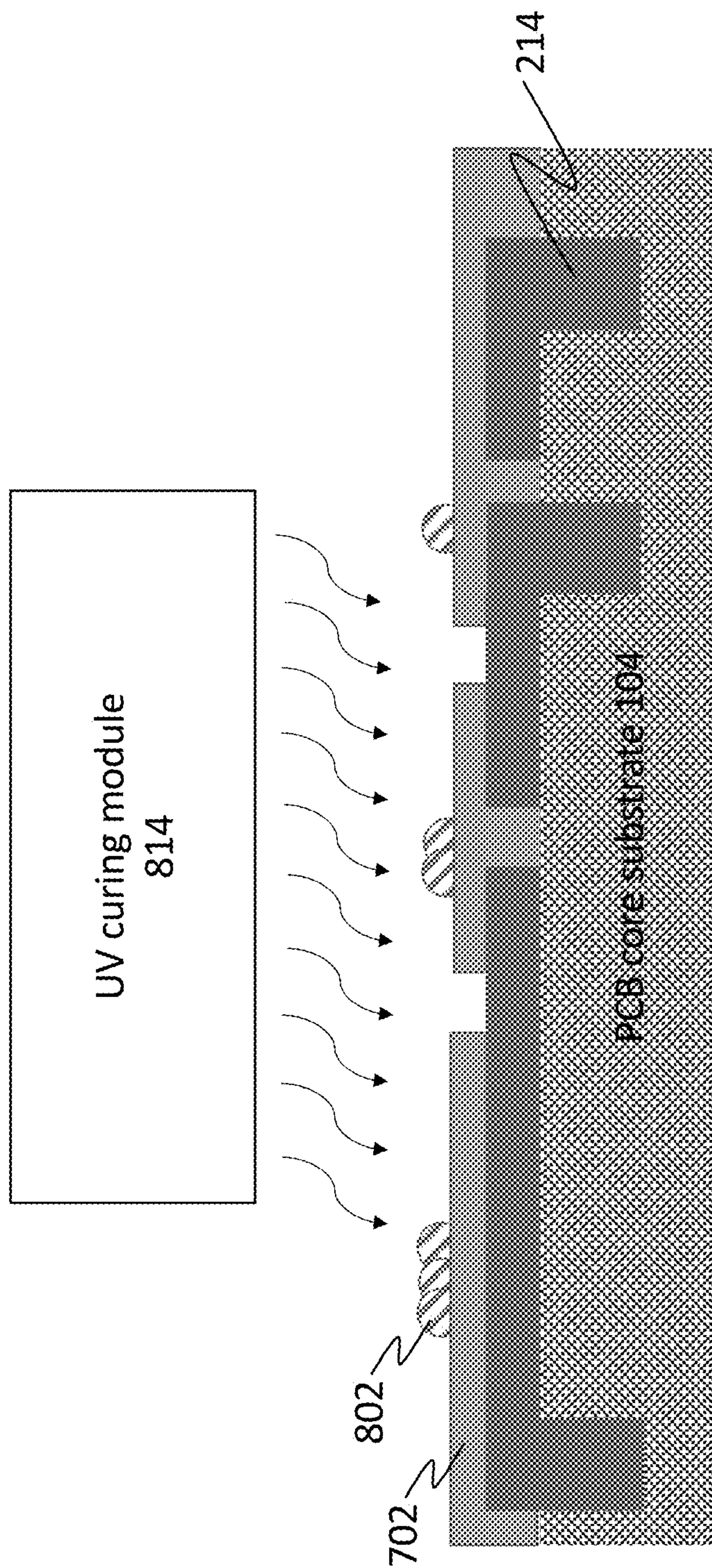


Figure 8b

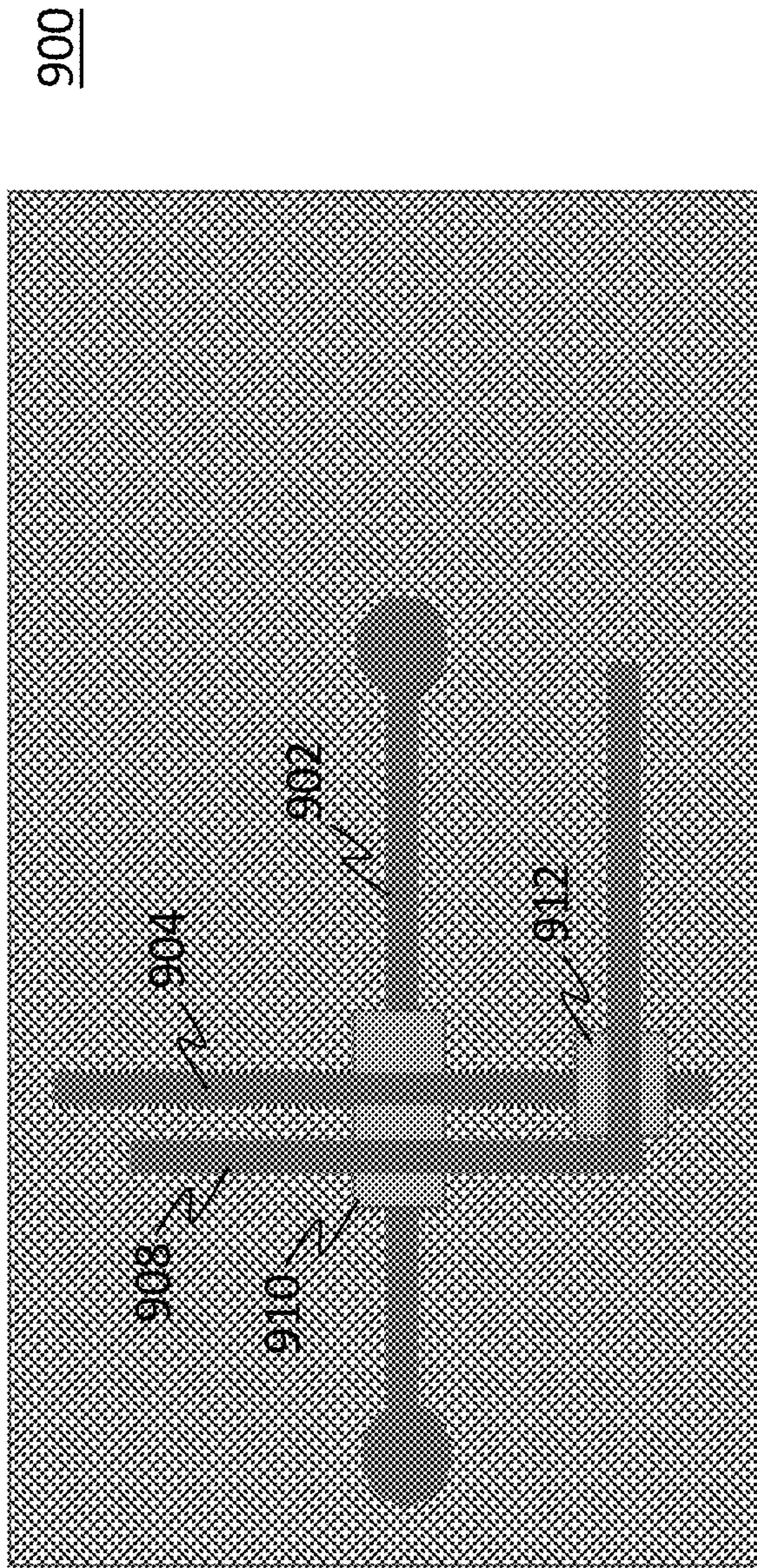
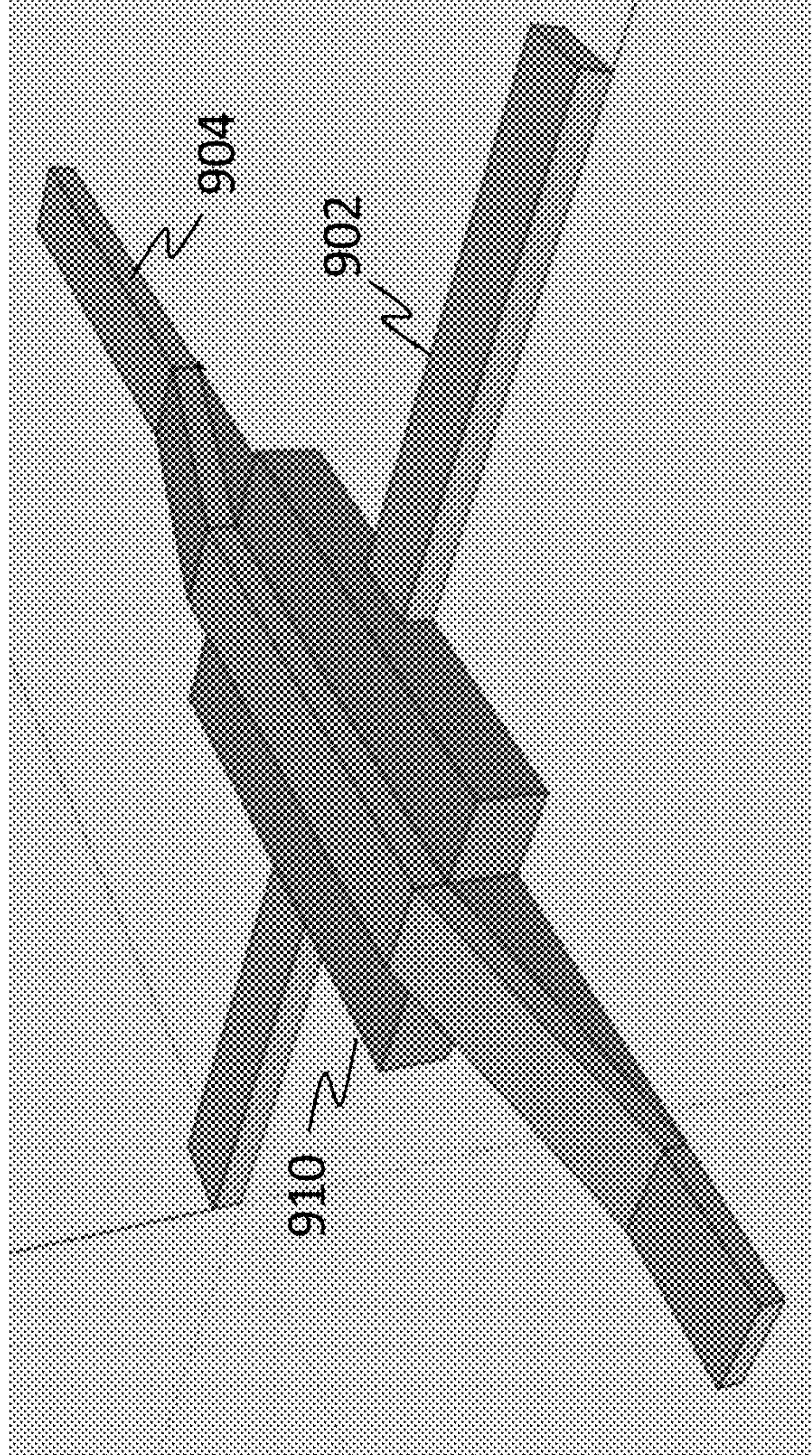


Figure 9a



900

Figure 9b

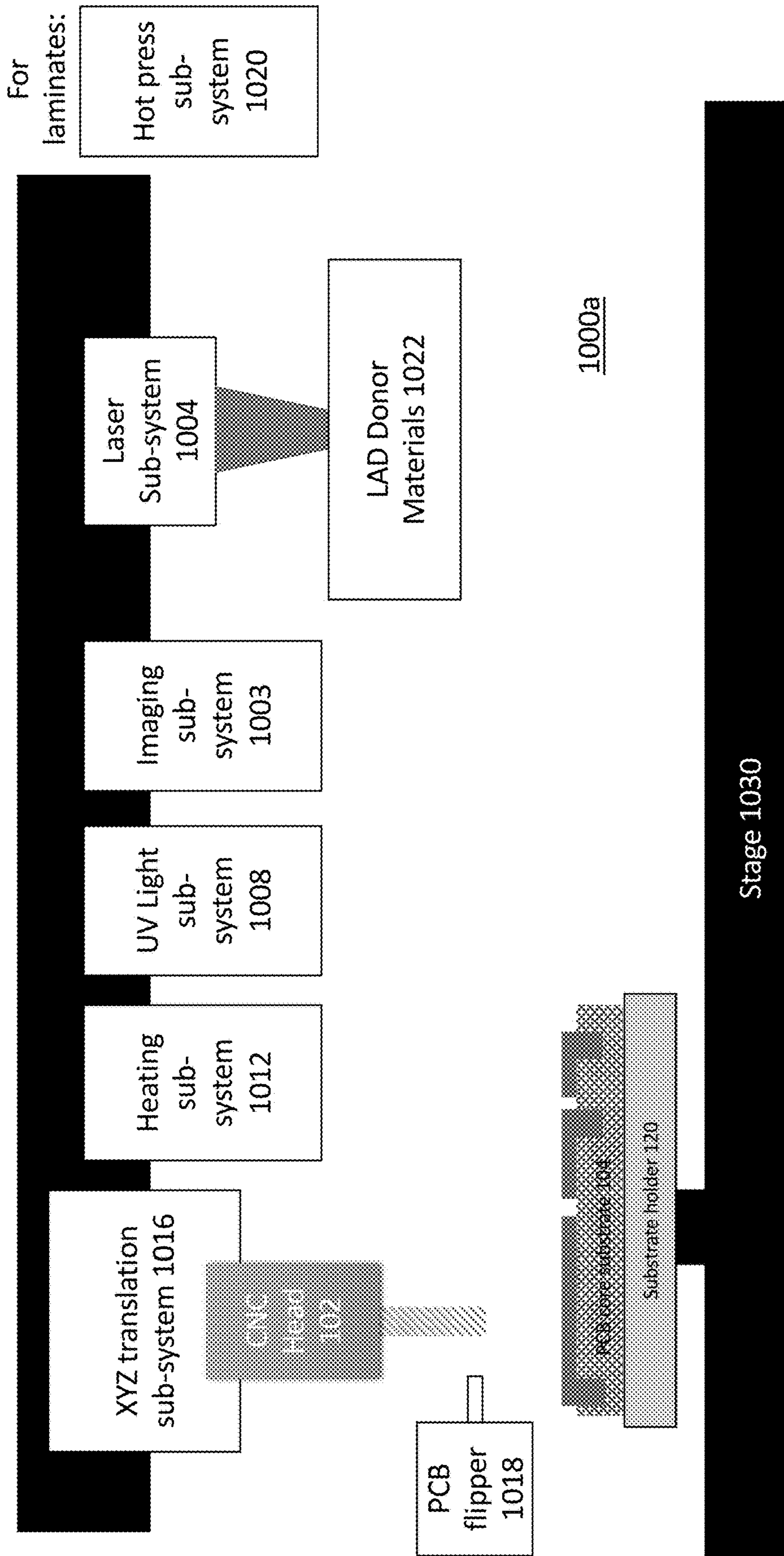


Figure 10a

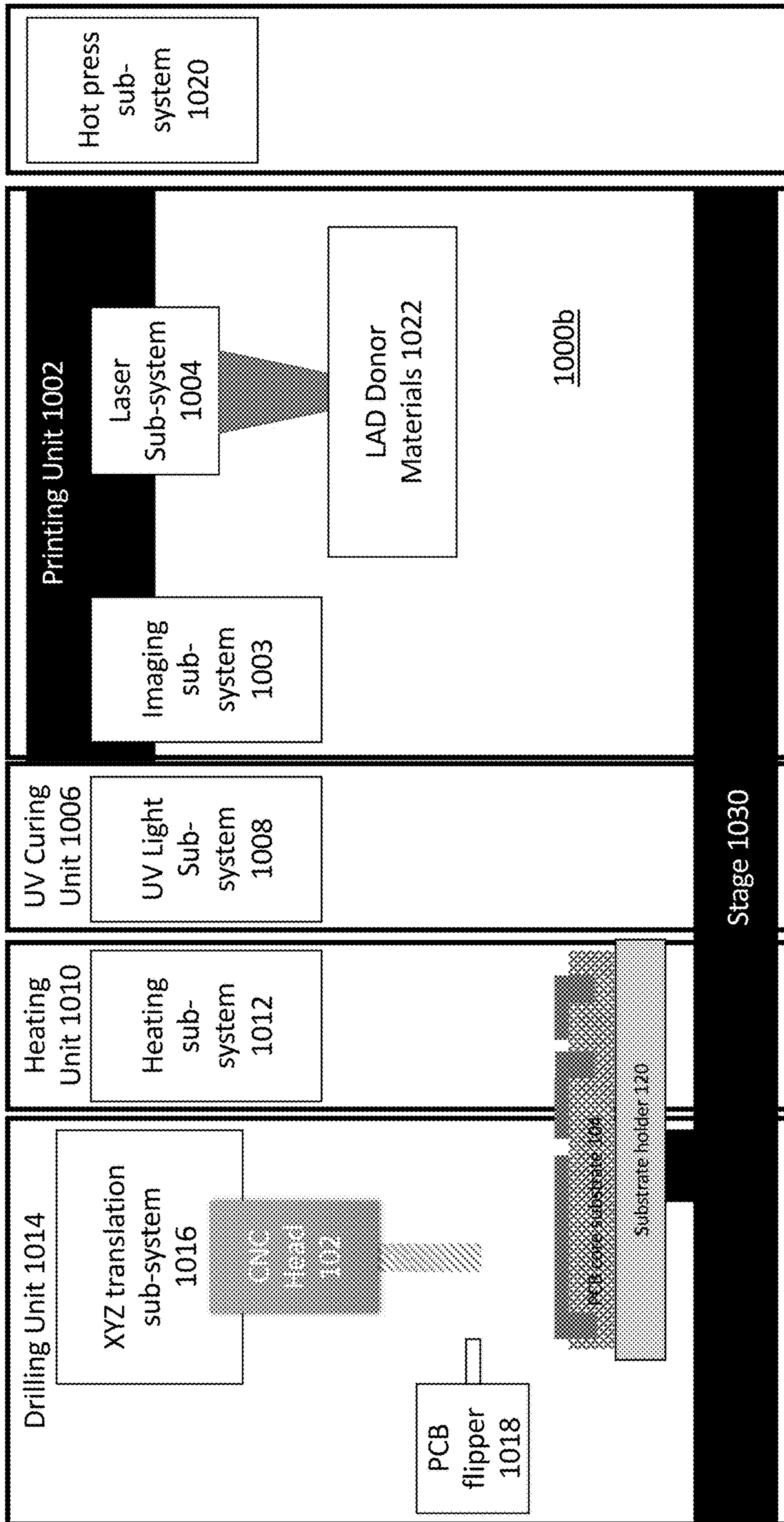


Figure 10b

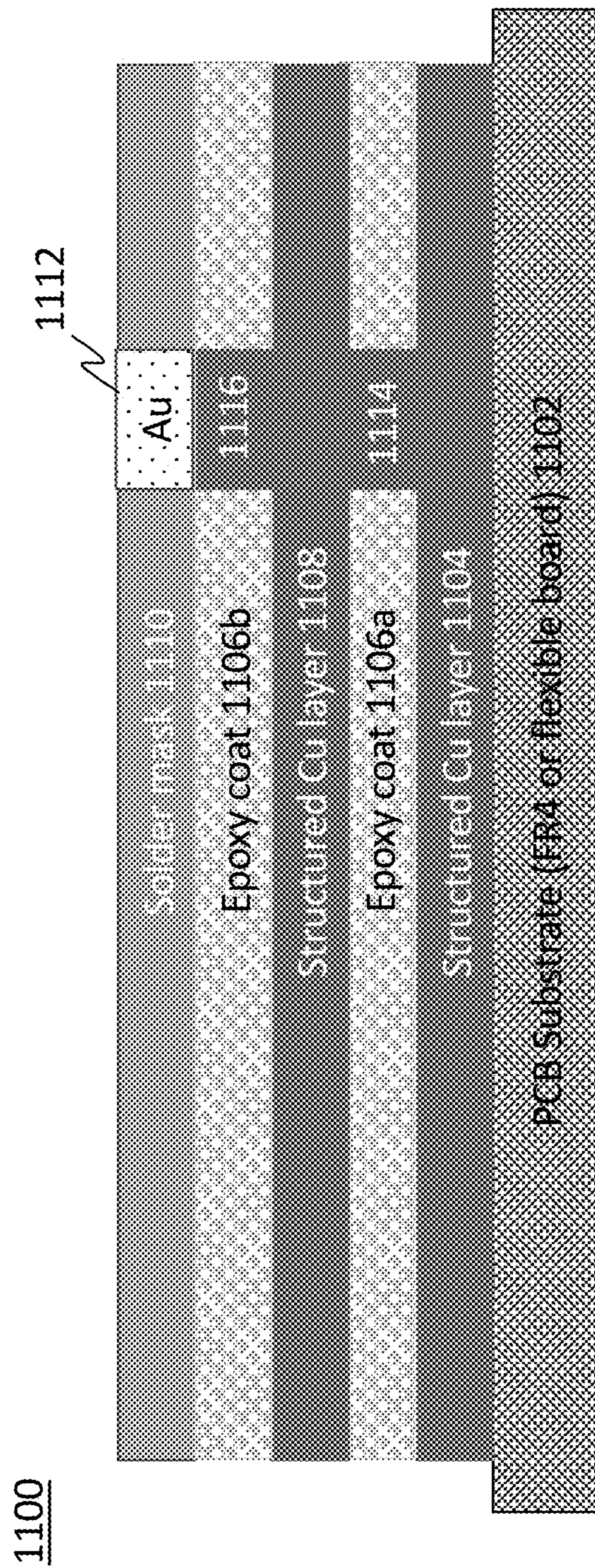


Figure 11a

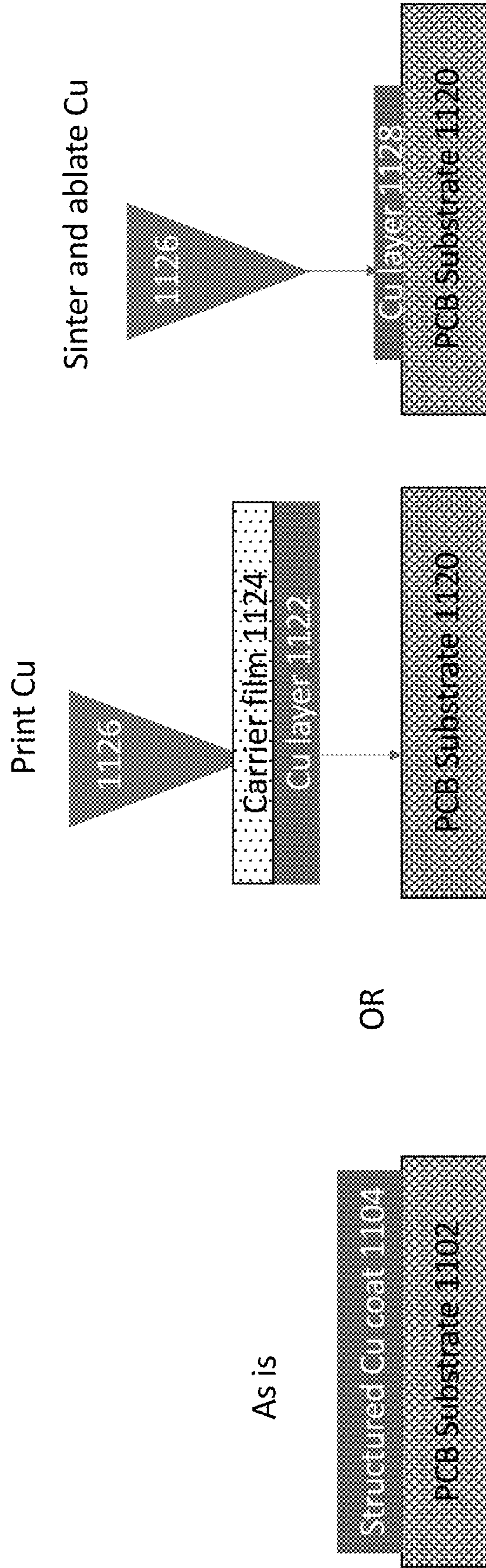


Figure 11b

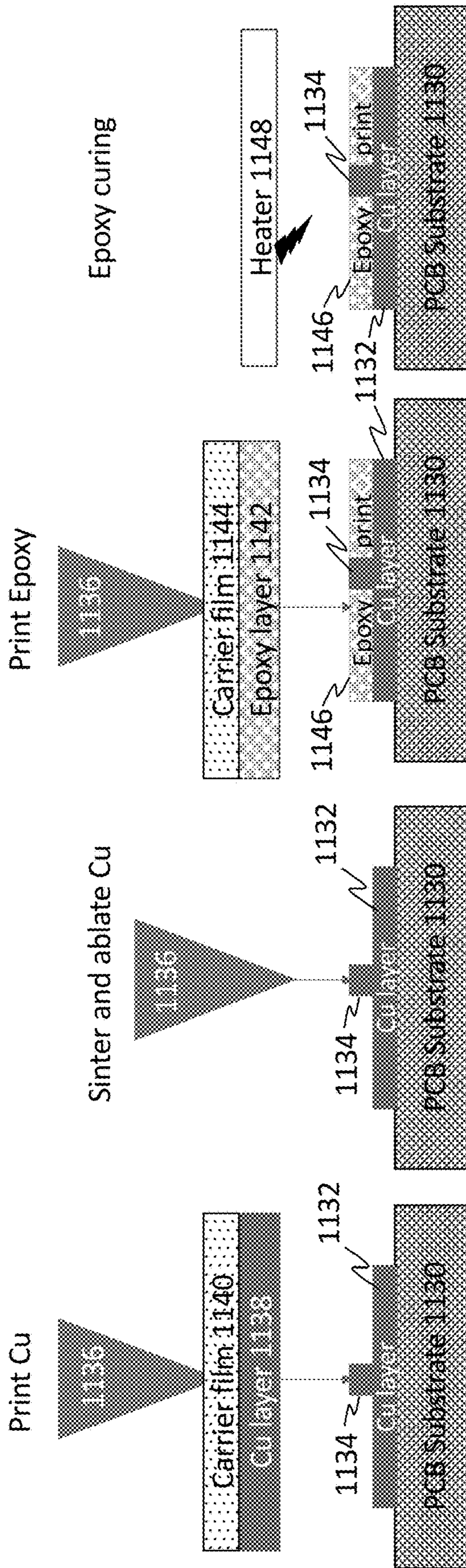


Figure 11c

Option 2: print Epoxy and heat, ablation for Cu lines, then print Cu and sinter

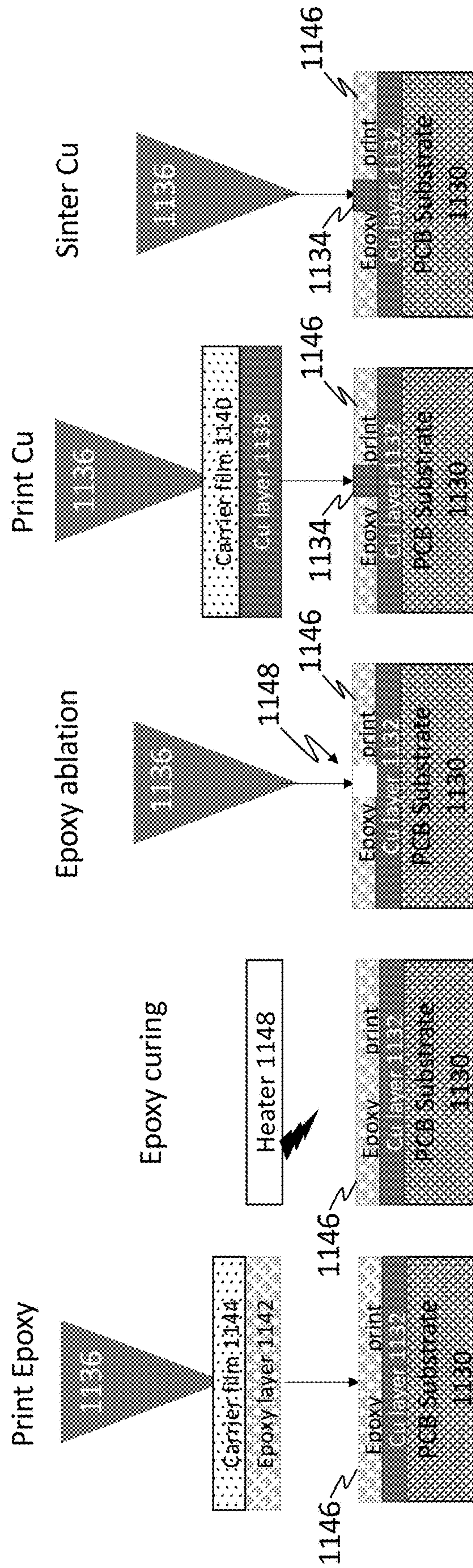


Figure 11d

1150



Figure 11e

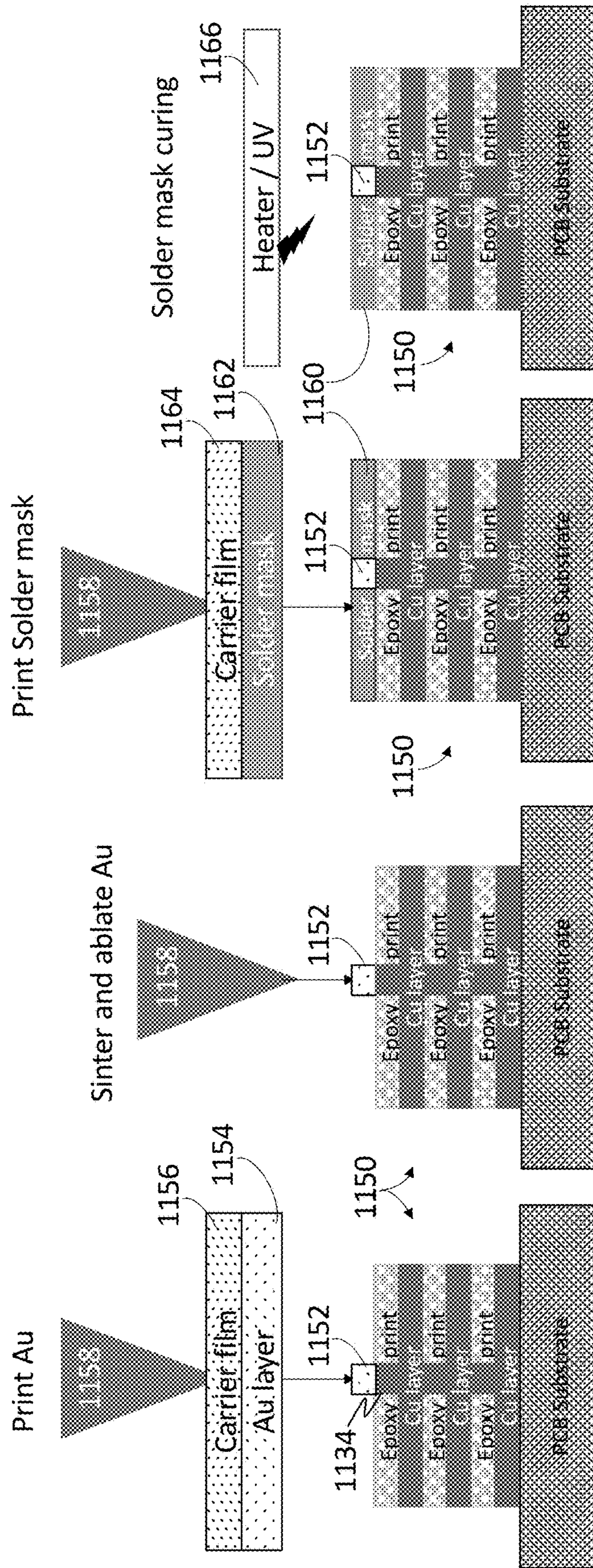


Figure 11f

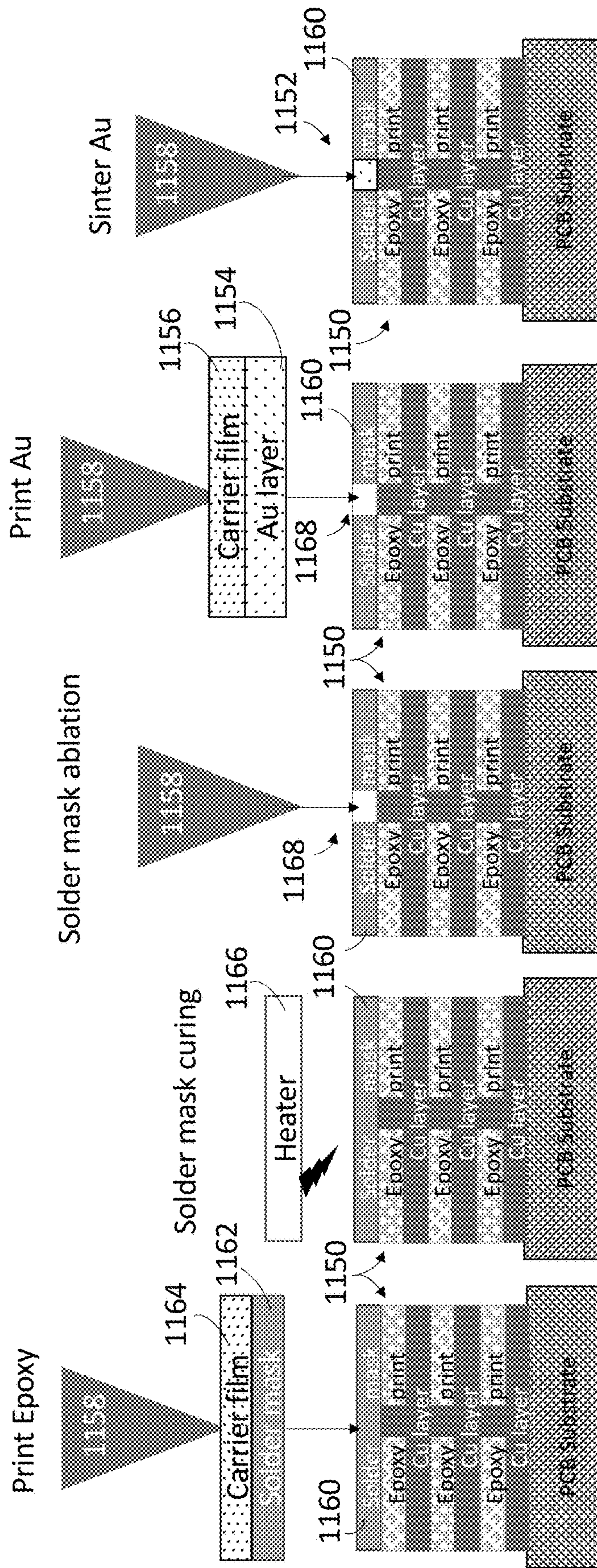


Figure 11g

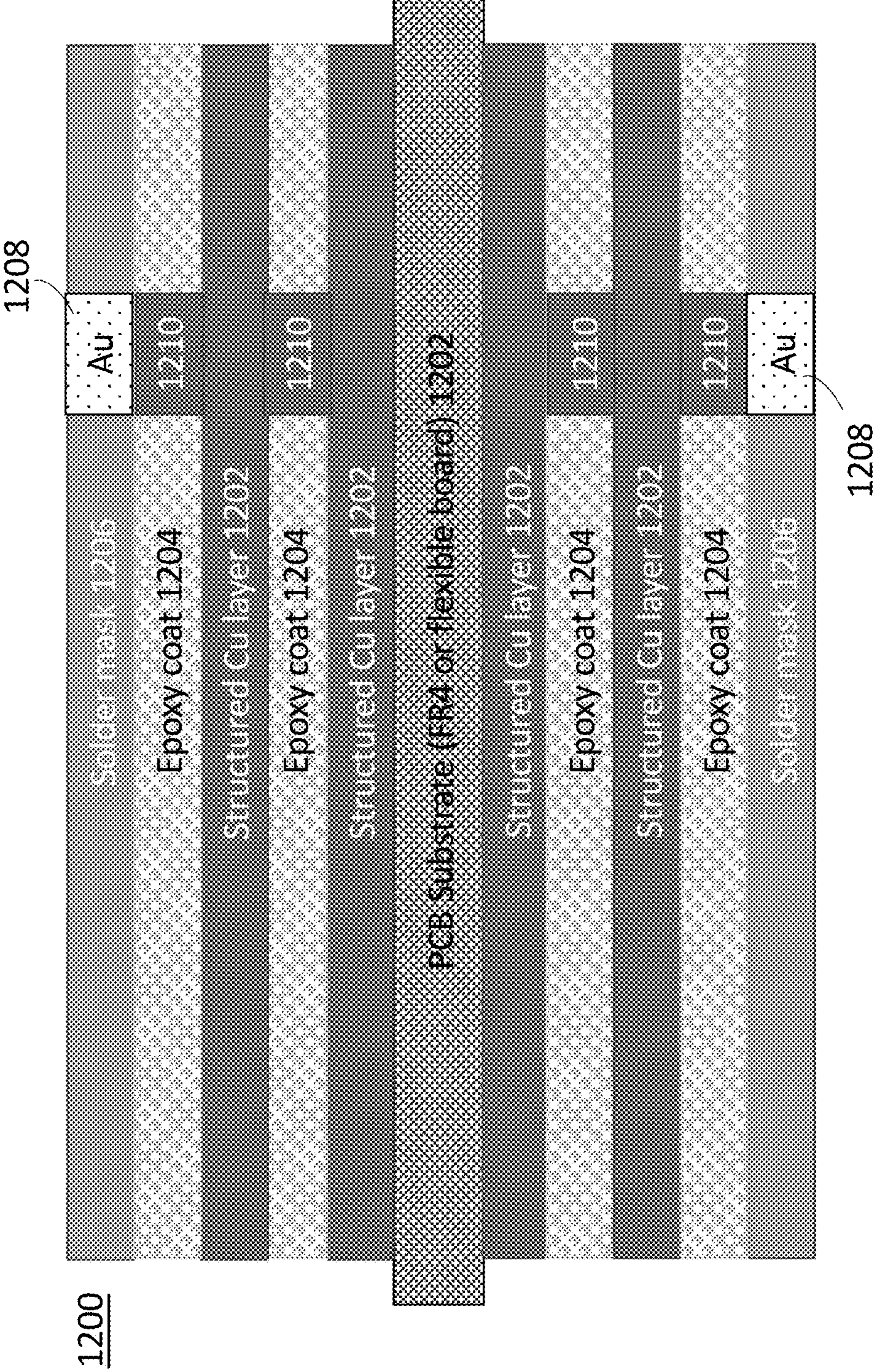


Figure 12a

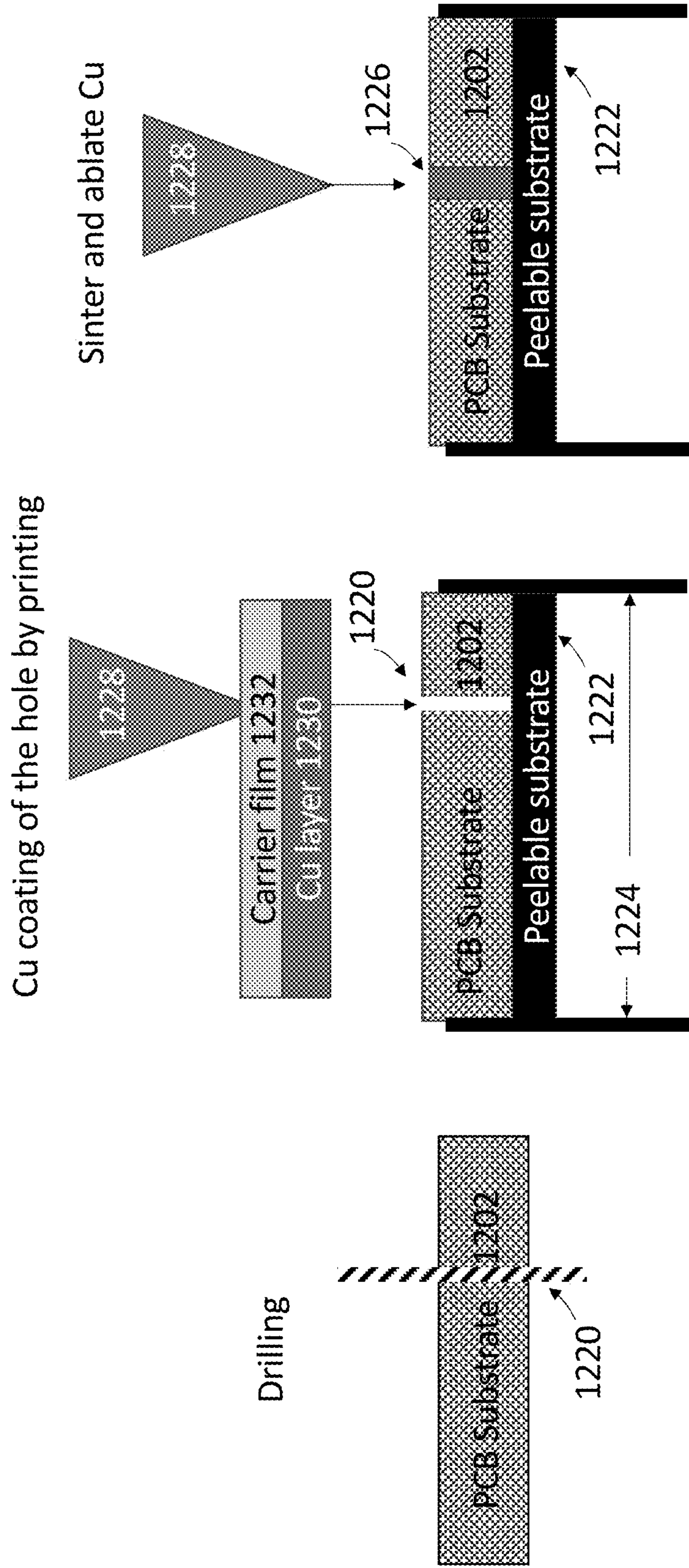


Figure 12b

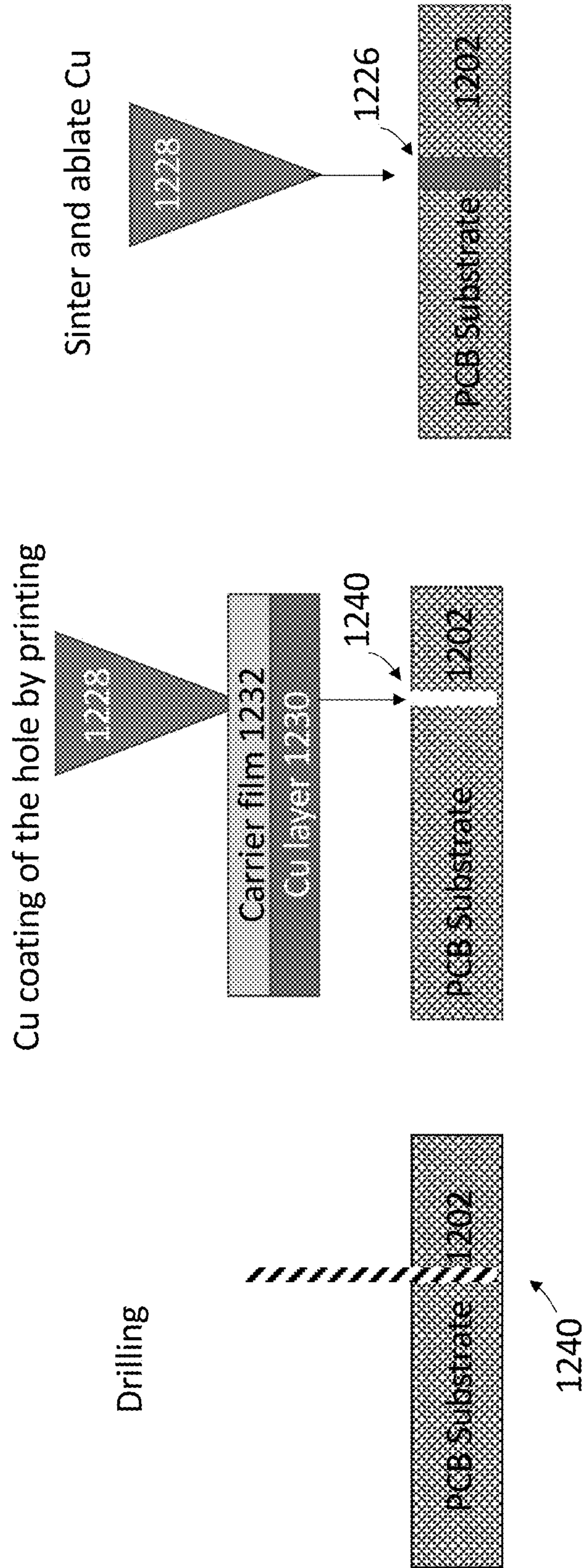


Figure 12c

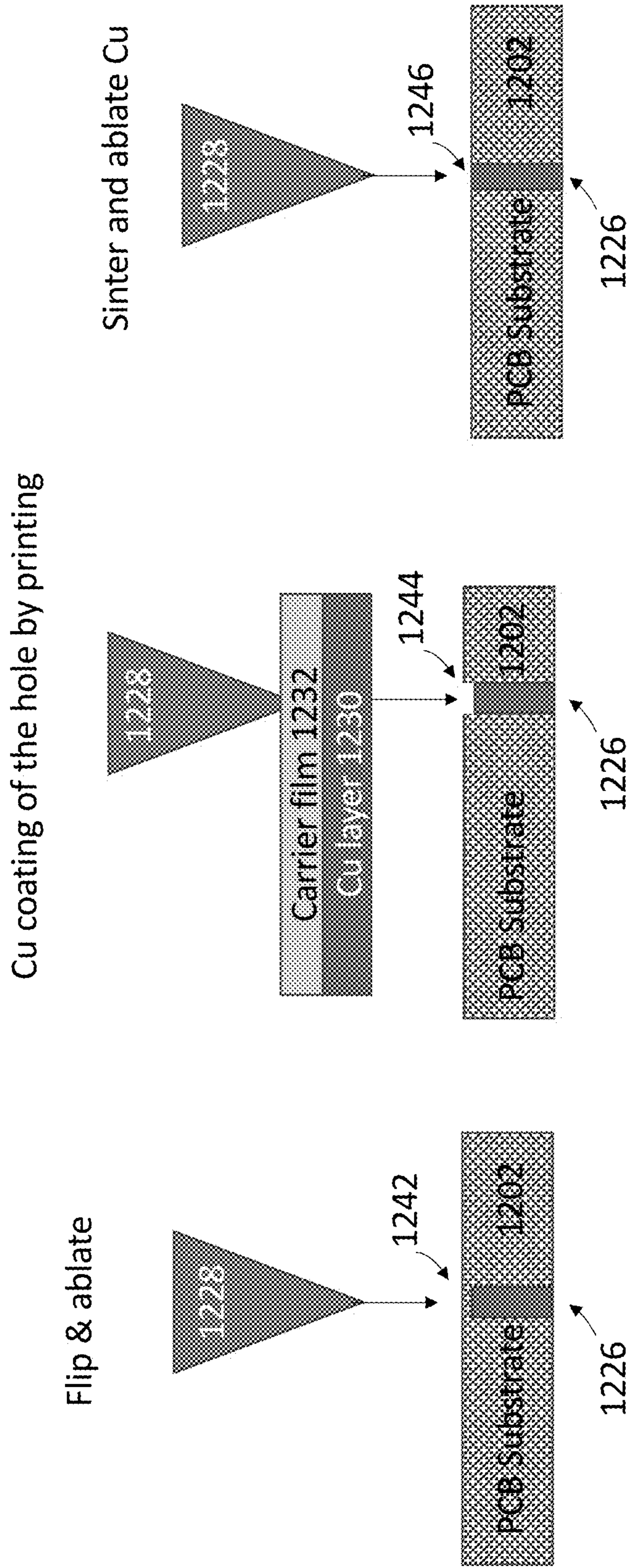


Figure 12d

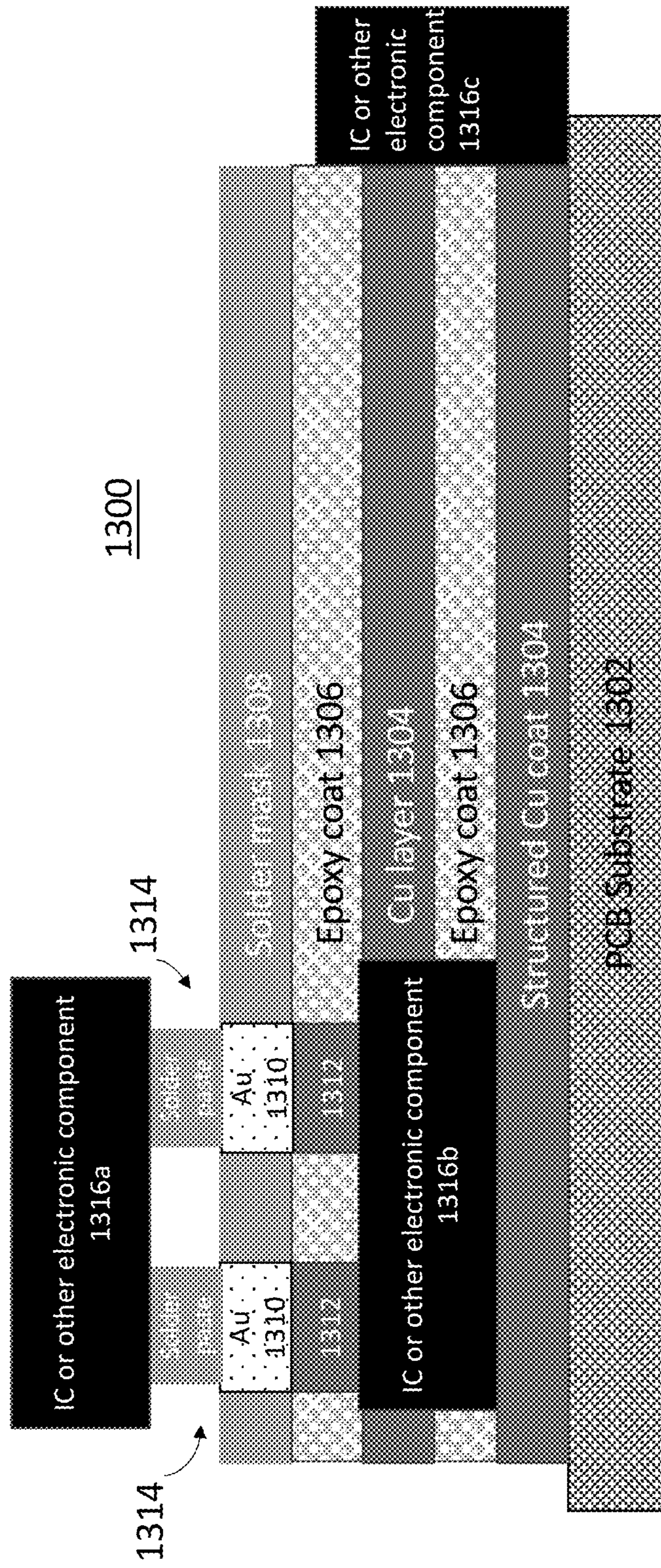


Figure 13a

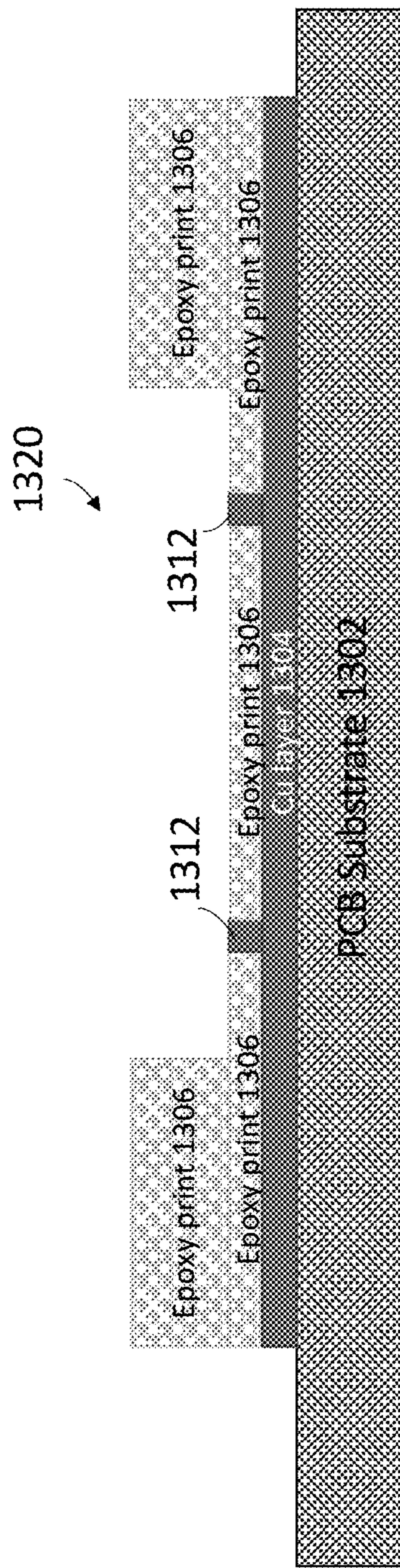


Figure 13b

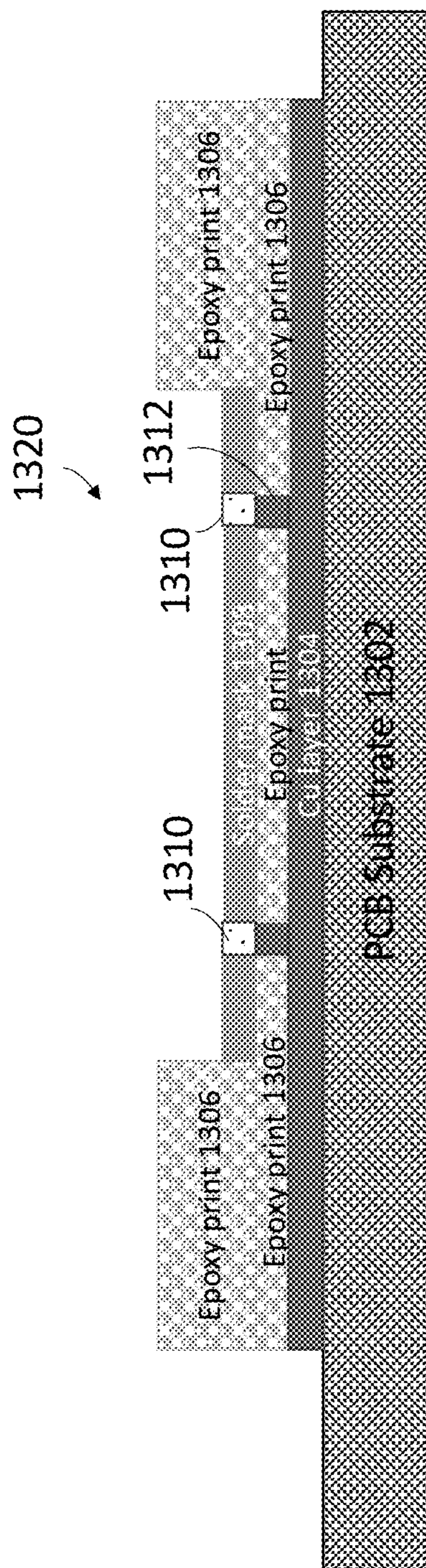


Figure 13c

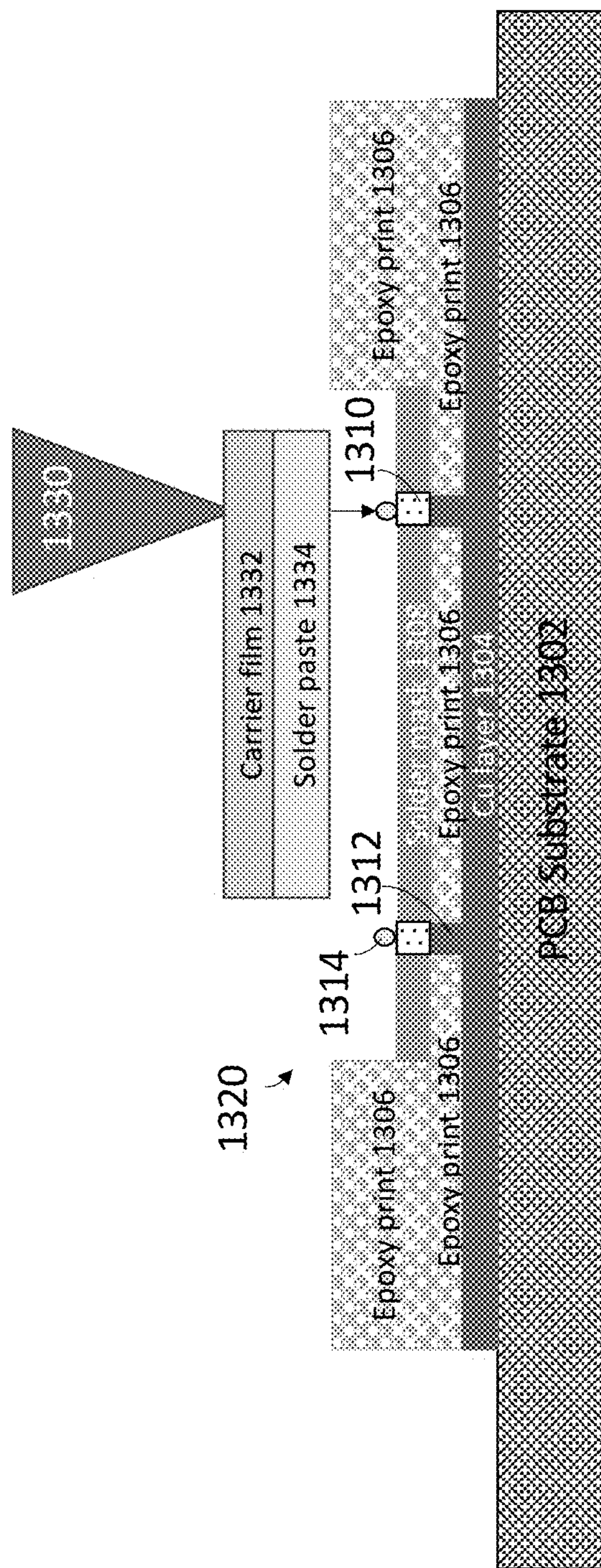


Figure 13d



Figure 13e

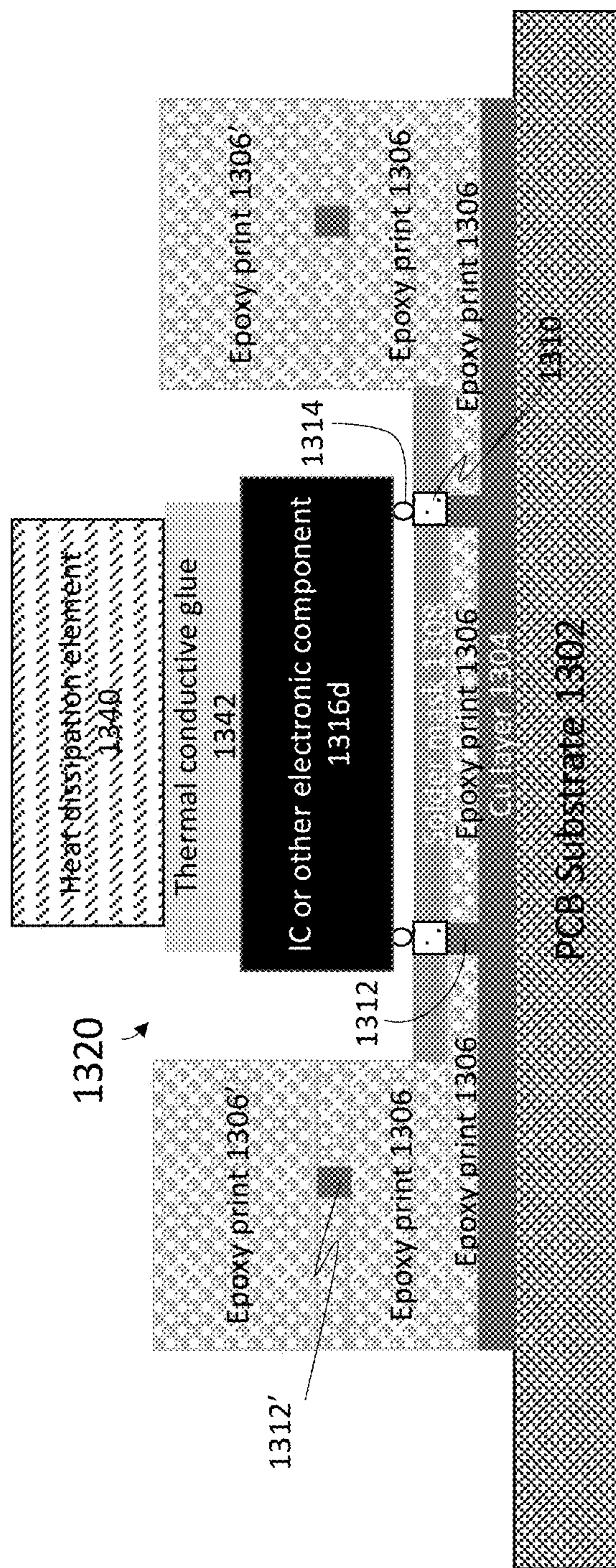


Figure 13g

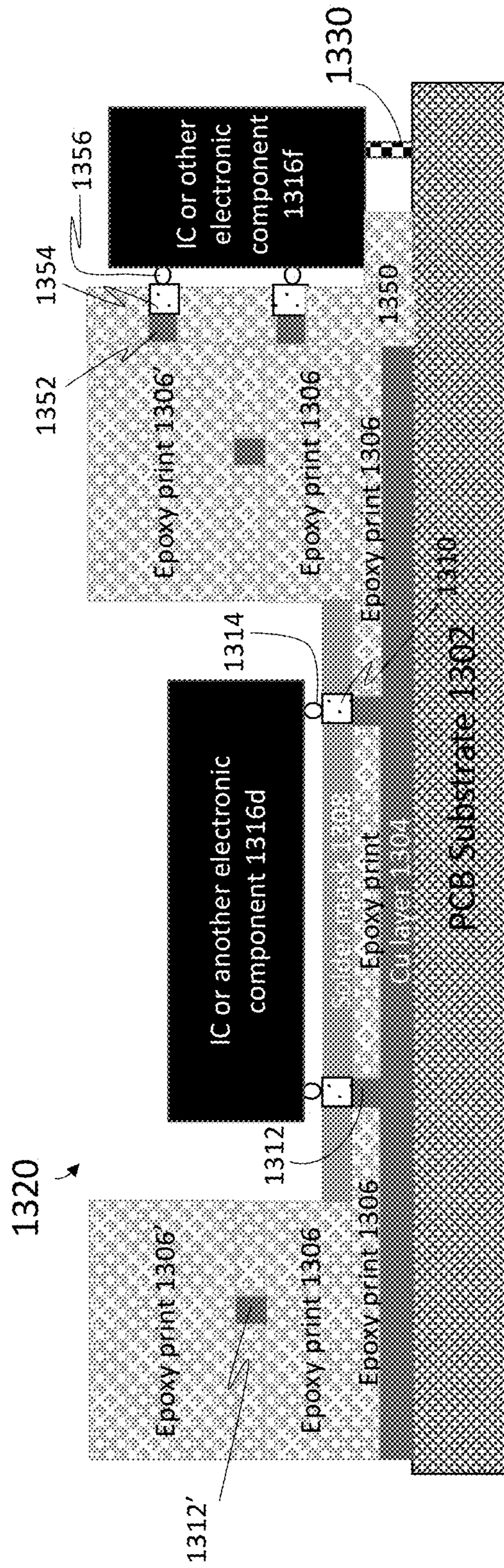


Figure 13h



Figure 13i

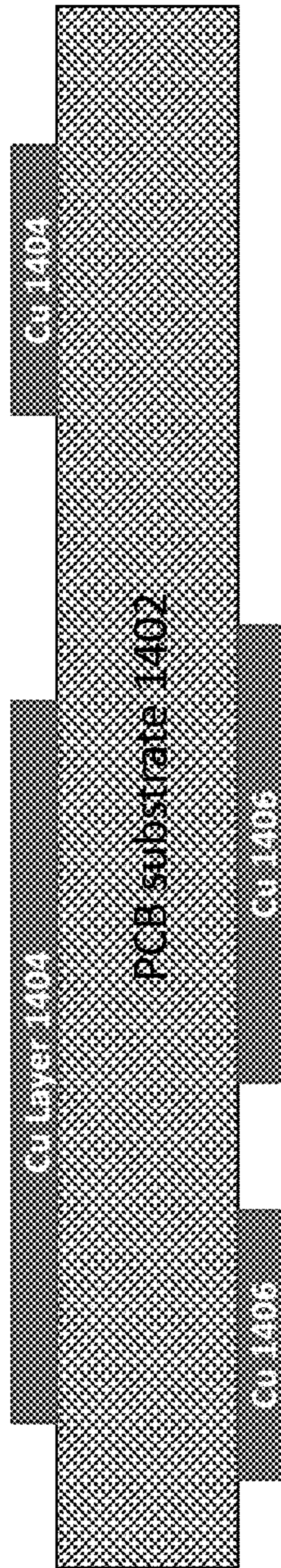


Figure 14a

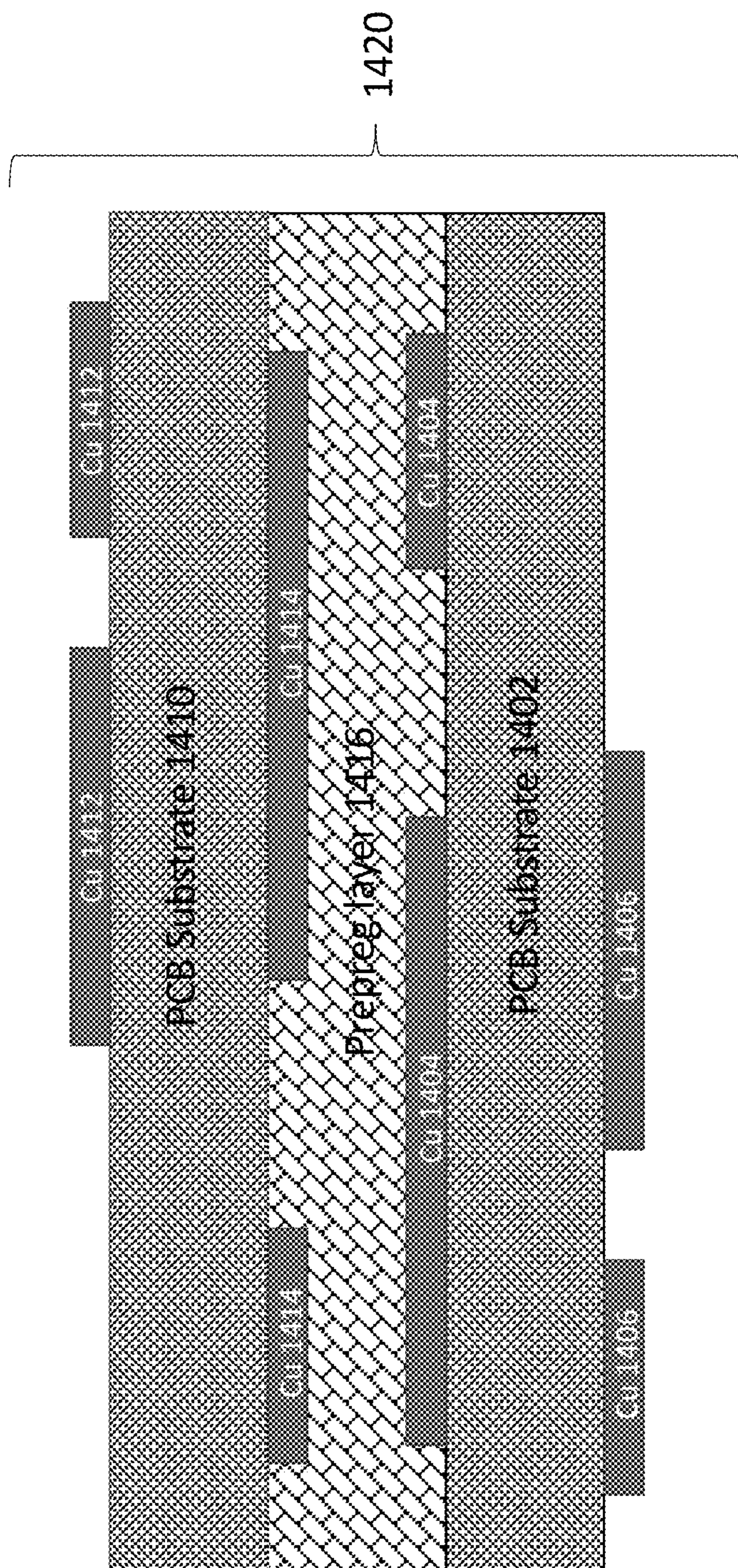


Figure 14b

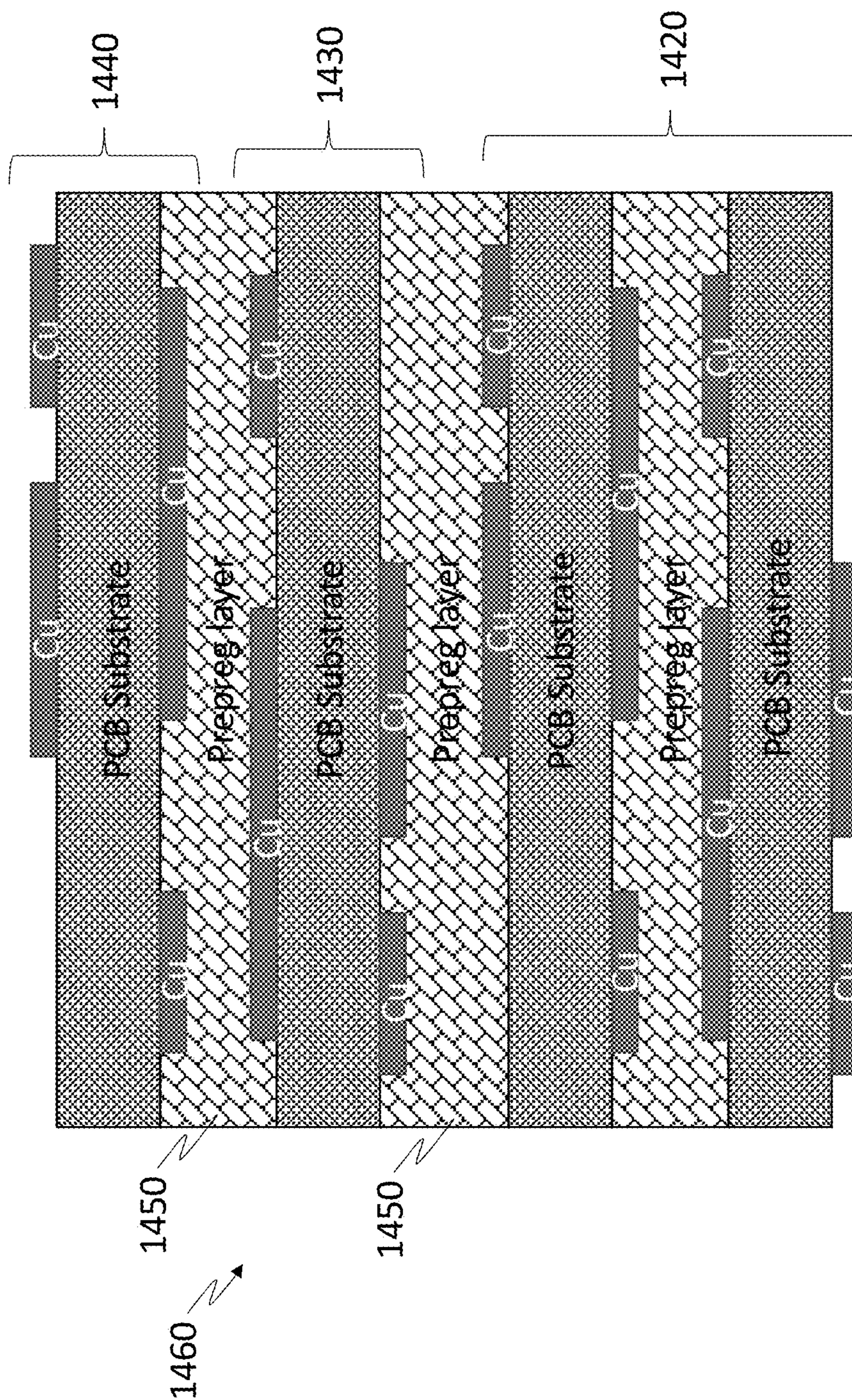


Figure 14c

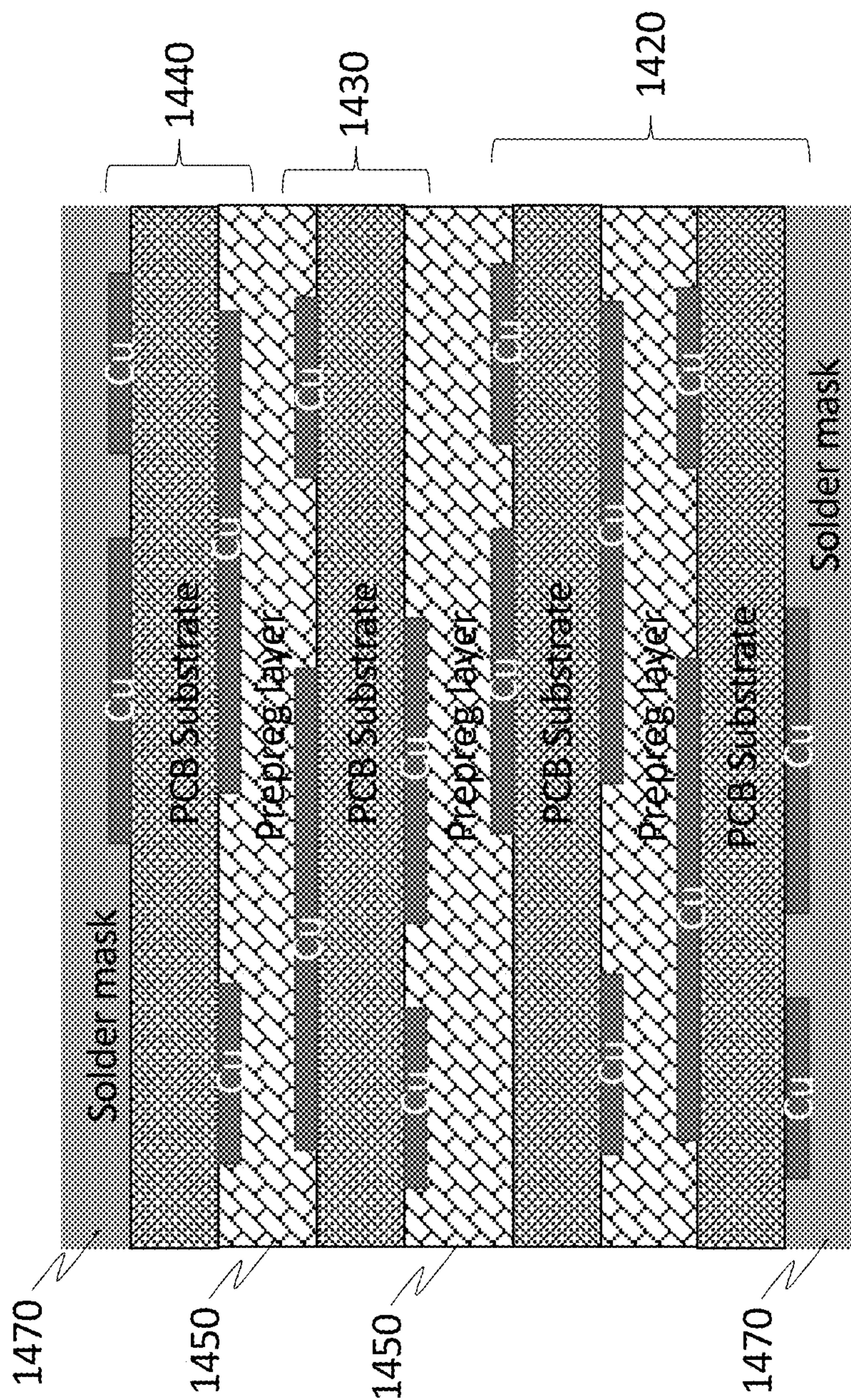


Figure 14d

PCB PRODUCTION BY LASER SYSTEMS

RELATED APPLICATIONS

This application is Divisional Application of U.S. patent application Ser. No. 17/249,217, filed 24 Feb. 2021 (now issued as U.S. Pat. No. 11,877,398), which is a NONPROVISIONAL of and claims priority to U.S. Provisional Application No. 63/200,034, filed 11 Feb. 2021, each of which is incorporated by reference herein.

FIELD OF THE INVENTION

The present invention relates to systems and methods for printing a printed circuit board (PCB) or a flexible PCB from substrate to full integration.

BACKGROUND

Surface Mount Technology (SMT) is an area of electronic assembly used to mount electronic components to the surface of a PCB as opposed to inserting components through holes in the PCB as in conventional assembly. SMT was developed to reduce manufacturing costs and allow efficient use of PCB space. As a result of the introduction of SMT and ever-increasing levels of automation, it is now possible to build highly complex electronic circuits into smaller and smaller assemblies with good repeatability.

The recent trend toward miniaturization creates a need for the fabrication of highly integrated PCBs. Printed circuit boards are generally fabricated by lithography using extractive methods, for example etching. Such a fabrication method provides formation of conductive lines by placing a conductive film on a substrate and etching away unnecessary portions of the conductive film to dissolution-remove a portion of the conductive film, where there are no circuits, with a corrosive solution and thereby to leave only necessary conductive lines. In addition, to improve integration, multi-layered printed circuit boards and double-sided printed circuit boards are required. Current fabrication of multi-layered printed circuit boards requires complicated processes including drilling to form through or via holes in order to enable conduction between multilayer boards, laminating the boards and soldering to adhere elements to the printed circuit board. When soldering is performed to adhere elements to the printed circuit board, a region, where a solder is melted and spread, is further required and the elements are thus located in an area wider than the size of elements themselves, which limits miniaturization. Therefore, there is a need for devices and methods enabling efficient and precise fabrication of complex circuit boards.

Flexible-rigid composite electronics represent a new generation of electronics, which can exhibit properties of both stretching as well as bending flexibility. These properties will afford electronic devices with conformity to bending and twisting as well as the capability to stretch and compress over a large strain scale. Because of their soft and conformable nature, stretchable electronics have shown great potential in biomedical engineering, e.g., epidermal electronic devices and implantable devices. As well as in the growing demand for wearable electronics, and other industries such as sensors, antennas with complex geometry, or radio frequency identification (RFID) tags to be placed on curved objects.

Progress in the field of flexible electronics is expected to play a critical role in a number of important emerging technologies. For example, flexible sensor arrays, electronic

paper, wearable electronic devices, and large area flexible active matrix displays. In addition, development of flexible integrated electronic systems and processing methods is also expected to significantly impact several other important technologies including micro- and nano-fluidics, sensors and smart skins, RFID, information storage, and micro- and nanoelectromechanical systems.

Flexible electronics refers currently to a technology for building electronic circuits by depositing electronic devices onto flexible substrates. Fabricating flexible electronics with performance that is equal to conventional (rigid) microelectronics built on brittle semiconductor wafers, but capable of optical transparency, being light-weight, stretchable/bendable formats, and being easy to print rapidly over large areas has been shown to enable diverse applications, such as flexible displays, thin film solar cells, and large area sensors and actuators. In all of these applications, the flexibility of both the circuits and the components incorporated on them represents important differences from typically rigid circuits. To date, it has proven to be a challenge to design a bendable (governed by Young's modulus, a modulus of elasticity describing a material property or parameter which is equal to a ratio between a mechanical tension and a corresponding elongation and thus a measure of the stiffness of a material) and stretchable (governed by Poisson's ratio, referring to the measurement of the relative change in width with a change in length, or the tendency of the component to "neck in" during stretching) electronics based on inorganic materials due to their small fracture strain (high Young's modulus and Poisson ratio of 1). A typical embodiment of flexible electronics are thin film inorganics adopted as semiconductors, conductors, and/or insulators on substrates to minimize strains induced by bending or stretching. Another embodiment is represented by circuits in wavy patterns, which can offer fully reversible stretchability/compressibility without substantial strains in the circuit materials themselves.

SUMMARY OF THE INVENTION

The present inventors have recognized that it is desirable to have a "one stop shop" for the production of PCB s; replacing a highly complex technology that presently utilizes approximately 20 stages to produce a single PCB board, not including electronic component placement on the board, by a machine that performs all the production stages and can even effect electronic component placement on the board.

Accordingly, the present invention relates to systems and methods for printing a PCB (rigid or flexible) from substrate to full integration. Various embodiments of the invention utilize a laser-assisted deposition (LAD) system to print a flowable material on top of a substrate by laser jetting to create a PCB structure to be used as an electronic device in a production line. In one embodiment, a system for PCB printing includes a jet printing unit, an imaging unit, curing units, and a drilling unit that print directly on a board substrate such as a glass-reinforced epoxy laminate material (e.g., FR4) or others. The jet printing unit can also be used for sintering and/or ablation of materials. Such a system can print copper and gold pastes, epoxies, and solder masks and cure them by heating or by ultraviolet (UV) radiation. Such a system can also print epoxy in joints of copper lines, e.g., where the epoxy layer is printed as a bridge on top of copper line on the PCB board. PCB boards produced according to the present systems and methods may be one-sided or double-sided.

In one embodiment, the present invention provides a method of fabricating of a PCB assembly in which a metal layer is deposited on a PCB substrate by LAD in which metal droplets from a donor substrate are jetted onto the PCB substrate and/or into one or more holes therein using a laser to form a layer of metal on the PCB substrate. The layer of metal is subsequently dried and sintered, and the jetting, drying, and sintering are repeated until the layer of metal reaches a desired thickness. Thereafter at least one passivation layer is formed over the layer of metal. If needed, the layer of metal may be ablated (e.g., using the same laser as was used for the deposition) if it exceeds the desired thickness. The passivation layer may be a layer of epoxy that is deposited or coated over the metal layer using a roller or blade. Or the passivation layer may be a layer of epoxy printed over the metal layer from an epoxy coat on a donor substrate by LAD. If needed, one or more additional metal layers and epoxy layers may be likewise printed using LAD. The layer of epoxy may be cured by hot air and/or infrared (IR) irradiation.

In some instances, the layer of metal will include a first metal trace, and the epoxy layer will include at least a first portion of epoxy that covers at least a first portion of the first metal trace. Additional metal layers printed over the epoxy layer may thus include a second metal trace having at least a portion disposed over the first portion of the epoxy layer that covers the first portion of the first metal trace. That is, the epoxy layer may form a bridge over which the second metal trace can cross the first metal trace on the same dies of the PCB substrate without causing a short circuit.

In some cases, the one or more holes in the PCB substrate will be formed in a first side of the PCB substrate by drilling or laser engraving the first side of the PCB substrate, but those holes will not pass through an entire thickness of the PCB substrate. Then, the layer of metal may be formed in the one or more holes by LAD from the first side of the PCB substrate; that is, the side on which the holes are present. Subsequently, the PCB substrate may be flipped and the one or more holes completed through the thickness of the PCB substrate by drilling or laser engraving through the second (reverse) side of PCB substrate. Thereafter, remaining portions of the one or more holes exposed by the drilling or laser engraving through the second (reverse) side of PCB substrate may be metalized by LAD in the fashion discussed above.

Where desired or needed, a solder mask layer is printed, e.g., by LAD, over the passivation layer and any intervening layers and/or components of the PCB assembly. Also, a label may be printed, e.g., by LAD, over the solder mask. As needed, holes and vias in the various layers, including the solder mask, are formed and metalized by LAD so as to provide electrical connections to the metal layers in the PCB assembly and to provide attachment points for electronic components.

In further embodiments, the present invention provides a method of fabricating of a PCB assembly that includes printing, by LAD, a metal onto an epoxy laminate, the metal being jetted as droplets from a metal coating or foil on a donor substrate by a laser into channels and/or holes created in the epoxy laminate by laser engraving, and subsequently attaching the epoxy laminate to a PCB substrate or a previously formed epoxy layer disposed over a PCB substrate by hot pressing, with the metal jetted into the channels and/or holes of the epoxy laminate being between the PCB substrate and an upper surface of the epoxy laminate in the portion of the PCB assembly. A solder mask layer may be printed, by LAD or otherwise, over the upper surface of the

epoxy laminate and any intervening layers and/or components of the PCB assembly, and a label printed on the solder mask.

In yet additional embodiments, the present invention provides a system for fabricating a PCB assembly in which a substrate holder, configured to hold a PCB substrate, is translatable between a plurality of processing stations, including a printing station configured for LAD of one or more materials (e.g., copper and gold pastes, epoxy, and solder mask material, etc.) by jetting respective ones of the materials individually from respective donor substrates on which the respective ones of said materials are coated or otherwise disposed, a curing station configured to cure by heating, IR irradiation, or UV irradiation, deposited ones of the materials on the PCB substrate, and a drilling station configured to drill or engrave through holes and/or vias in the PCB substrate and/or layers of ones of the materials disposed thereon. The printing station may also be configured for laser sintering and/or laser ablation of the respective ones of the materials printed on the PCB substrate and/or additional layers of said PCB assembly. A unit configured to flip the PCB substrate to allow access to both sides of the PCB substrate by the printing station, curing station, and/or drilling station may also be provided.

In yet another embodiment, the present invention provides a method for fabricating a printed circuit board PCB assembly in which one or more vias are drilled or laser engraved in a PCB substrate from a first side of the PCB substrate. The vias do not extend through an entire thickness of the PCB substrate. A metal paste is deposited, by LAD, over at least a first portion of the PCB substrate and into one or more of the vias to a first thickness. The depositing is performed by jetting small volumes of metal paste from a donor film on a first carrier substrate by an incident laser beam onto the PCB substrate and into the one or more vias, curing the metal paste deposited on the PCB substrate and into the one or more vias, sintering the deposited and cured metal paste using a same laser that was used for depositing the metal paste, and repeating the depositing, curing and sintering of the metal paste, thereby forming successive thicknesses thereof on the PCB substrate and in the one or more vias, until a desired thickness of the metal paste on the PCB substrate and in the one or more vias is reached. Then a passivation layer is printed by LAD on the desired thickness of metal paste on the PCB substrate and in the one or more vias; the printing being performed by jetting small volumes of epoxy from a second carrier substrate using the same laser that was used for depositing the metal paste, and curing the passivation layer. Finally, a solder mask is formed by LAD over the passivation layer, with the formation involving jetting of small volumes of solder mask material from a third carrier substrate using the same laser that was used for depositing the metal paste, and curing the solder mask. The solder mask layer, passivation layer, and metal paste layer(s) may be cured using heat, or IR or UV radiation.

In various instances, the PCB substrate is moved between the drilling, the depositing, the priming, and the forming processes on a stage that is translatable between positions at which the drilling, the depositing, the printing, and the forming processes take place. Also, the processes of depositing the metal paste and printing the passivation layer may be performed multiple times prior to forming the solder mask so as to produce a PCB assembly having multiple layers of both metal paste and epoxy between the PCB substrate and the solder mask (e.g., on one or both sides of the PCB substrate). Metal electrical connectors for an elec-

tronic component may be formed within the passivation layer and/or the solder mask and the metal electrical connectors may be formed from different metals (e.g., Cu, Au, Ag, etc.). One or more electronic components may be attached to the metal electrical connectors, e.g., by one or more solder joints and in some instances one or more additional passivation layers and/or solder mask layers may be formed over the electronic component. Where needed, support structures for the electronic components may be formed by LAD prior to attaching the electronic components. Also, as discussed above, epoxy bridges may be used to avoid short circuits between different layers of electrical traces.

These and further embodiments of the invention are described in greater detail below.

BRIEF DESCRIPTION OF THE DRAWINGS

The present invention is illustrated by way of example, and not limitation, in the figures of the accompanying drawings, in which:

FIGS. **1a-1d** illustrate, in a conceptual manner, different processes for creating a via in a PCB.

FIGS. **2a-2c** illustrate schematically a process of printing a metal layer into a via in accordance with embodiments of the present invention.

FIG. **2d** illustrates one example of printing on the edges of a via to create a metal coat on the via walls in accordance with embodiments of the present invention.

FIGS. **3a** and **3b** illustrate various alternatives for the metal printing process stages.

FIGS. **4a** and **4b** illustrate various ways to print an epoxy layer onto the surface of a PCB, by a roller or blade (FIG. **4a**) or by direct printing (FIG. **4b**), in accordance with embodiments of the present invention.

FIG. **4c** illustrates the heating of a PCB produced according to the methods illustrated in FIGS. **4a** and **4b** to achieve the final laminate properties.

FIGS. **5a-5c** illustrate additional methods to create a PCB by engraving solid laminate (FIG. **5a**), printing a metal layer onto the laminate (FIG. **5b**), and drying and pressing against the former layers (FIG. **5c**).

FIGS. **6a-6c** illustrate yet additional methods to create a PCB by engraving solid laminate after pressing against the former layers (FIG. **6a**), and then printing the metal layer onto the laminate (FIG. **6b**) and drying (FIG. **6c**).

FIGS. **7a-7c** illustrate printing of a solder mask onto a PCB (FIGS. **7a-7b**) and curing thermally or by UV light (FIG. **7c**).

FIGS. **8a** and **8b** illustrate printing of a label onto a PCB (FIG. **8a**) and curing thermally or by UV light (FIG. **8b**).

FIGS. **9a** and **9b** illustrate aspects of how the methods and systems of the present invention can eliminate the need for a double-sided PCB by using epoxy bridges on a single side of a PCB, thereby reducing PCB processing time.

FIGS. **10a** and **10b** illustrate aspects of a system for full production of PCBs in accordance with embodiments of the present invention.

FIGS. **11a-11g** illustrate aspects of a production cycle of a multilayer PCB, from substrate to a final board, in accordance with embodiments of the present invention.

FIGS. **12a-12d** illustrate aspects of a production cycle of a double-sided multilayer PCB board from substrate to a final board, in accordance with embodiments of the present invention.

FIGS. **13a-13i** illustrate aspects of a production cycle of a PCB with embedded electronics, in accordance with embodiments of the present invention.

FIGS. **14a-14d** illustrate aspects of a production cycle of a PCB in which laser assisted deposition is used for some aspects of PCB fabrication.

DESCRIPTION OF THE INVENTION

PCB production is a highly developed field with a significant number of stages. The current invention is aiming at simplifying the production process through provision and use of a single system with several sub modules that can construct a PCB from the prepreg and laminate level up to a fully developed board with embedded electronics as a one-shop-stop. Systems and methods configured in accordance with the present invention need not necessarily employ different materials that are used today for conventional PCB production (although such new materials may be used), but may instead use those same materials in new and different ways. Hence, in various embodiments the present invention relates to systems and methods for printing PCB or a flexible PCB, from substrate level to full integration. As described below, embodiments of the invention may utilize LAD systems to print any flowable material on top of a substrate by laser jetting to create a PCB structure to be used as an electronic device in a production line.

The first stage in the PCB production involves drilling vias in the board substrate. As illustrated in FIGS. **1a-1d**, any of several approaches can be taken. For example, and referring to FIG. **1a**, in a PCB board production system **100a** a CNC machine **102** may be used for drilling into a PCB core substrate board **104**. As indicated, the PCB core substrate board **104** may be made of a glass-reinforced epoxy laminate such as FR4. Modules of this kind, including those with an additional cleaning unit for removal of drilling waste, are used widely today. The CNC head **102** is mounted on an stage (not shown) that is moveable in three dimensions, and the substrate board **104** is held flat from its sides or on a sacrificial substrate (not shown in this view). The drilling head **102** creates holes **106**, optionally with different diameters, at desired locations on the board **104**. In the current system it is beneficial to drill only vias and not through holes because a laser can be used to remove extra material after the board is flipped, simplifying the placement of copper paste in the holes and keeping the substrate holder clean (through holes can contaminate the holder surface).

As explained below, other aspects of the PCB board production system are laser-based. Therefore, rather than using a CNC drill head, all of the engraving or cutting involved in the production of a PCB, including the formation of vias, can be done using the laser. To that end, FIG. **1b** illustrates an embodiment of a PCB board production system **100b** that engraves the vias using a laser system **110**. When laser **110** is used, a power or light intensity meter **112** can be used to evaluate the hole depth. By measuring the light intensity at the location of a via during laser drilling, the via hole depth can be evaluated (e.g., by table lookup or other means) so that the drilling is stopped before the hole passes through the entire thickness of the board substrate **104** (e.g., so that a thickness “z” of the substrate remains). The light intensity measured may be that of the cutting laser beam **114** itself or of a separate light source (e.g., one positioned above the cutting position).

As mentioned, drilling/cutting only vias and not through holes simplifies the cleaning of a substrate holder. However, referring to the PCB board production system **100c** shown in

FIG. 1c, another solution that may be employed is the addition of a peelable substrate **108** beneath the PCB substrate **104**. For example, the peelable substrate **108** may be added to one side of the PCB substrate **104** and, after drilling and cleaning the opposite side, the peelable substrate **108** may be removed to allow for drilling of the side of PCB substrate **104** on which it was originally placed. Where the peelable substrate **108** is used, an appropriate coating and peeling unit is added to the system to enable these operations.

Referring now to FIG. 1d, still another way to keep a sample holder **120** of a PCB board production system **100d** clean is to use a sacrificial layer **116** (e.g., made of plastic or a similar material) between the sample holder **120** and the PCB substrate **104**. In this case, the sacrificial layer **120** should be able to hold the PCB substrate **104** flat. Typically, the sample holder **120** is a vacuum holder that keeps the PCB substrate **104** flat with the holder surface, but if a sacrificial layer **116** is added it should be able to maintain the vacuum on the PCB substrate **104**. Accordingly, some holes **118** must be added in the sacrificial layer **116** so that vacuum is maintained against the PCB substrate **104**, and these holes should be located away from the drilling hole **118** positions. For each application, a new sacrificial layer **116** must be designed (to account for the locations of the drilling holes), and it can be used many times.

Turning now to the metallization process, in embodiments of the present invention this may be performed by paste printing. FIG. 2a illustrates how, in a PCB board production system **200**, a non-flat surface, such as a PCB substrate **104** in which vias **106** have been engraved/drilled, is covered by a metal paste **202** by laser induced jetting. Laser induced jetting is a form of LAD in which a laser beam **204** is used to create a patterned surface by controlled material deposition. In particular, laser photons provide the driving force to catapult a small volume of material **206** from a donor film **208** toward an acceptor substrate such as PCB substrate **104**. Typically, the laser beam **204** interacts with an inner side of the donor film **208**, which is coated onto a non-absorbing carrier substrate **210**. The incident laser beam **204**, in other words, propagates through the transparent carrier substrate **210** before the photons are absorbed by the inner surface of the film **208**. Above a certain energy threshold, material **206** is ejected from the donor film **208** toward the surface of the PCB substrate **104** which is situated on a stage (not shown in this view) in a work area.

Once deposited on the PCB substrate **104**, including in vias **106**, the metal paste **202** is dried by hot air **212**, see FIG. 2b, or by heating using an infra-red (IR) lamp or similar arrangement and the resulting metal film **214** can be sintered using a laser beam **216** that is passed over the deposited metal paste to produce a highly conductive metal (e.g., copper) film as shown in FIG. 2c. The same laser that was used for the paste deposition may be used for the sintering and can be used also for ablating deposited material that was not placed correctly—an inline repair to increase robustness.

Because the printing of the conductive film is an intermediate step, it is desirable that the formation of this layer does not take a long period of time. Accordingly, the material from which the conductive film is formed should take only a short time to cure (whether by IR irradiation, hot air, or both) and should not shrink much (if at all) during the curing process. Materials that take an excessive time to cure will impede the overall speed of the process, and those that shrink (at least more than a little bit) during curing will impart mechanical stress on the PCB substrate, which may lead to failure.

The active or conductive material used for the conductive film may comprise one or more metals. Metals that are contemplated include pure metals, metal alloys, and refractory metals. Copper is a common choice for PCB metallization, and may be used in embodiments of the present invention. The active material may be applied (printed) using LAD either from a solid state, e.g., small metal particles that are deposited on a plastic film can be used in the LAD process to generate a conductive layer, or in the form of a paste carried on a donor film as described above. The conductive film should be applied in an amount sufficient to fully support the subsequent electronic connections. This may mean applying several layers of paste, one atop the other, with curing steps after each application of a layer. As shown in FIG. 2d, the laser jetting process is highly accurate and so can be used to print the metal paste **208** directly on edges **220** of the vias **106**, enabling a way to coat only the vias holes and not to fill them. This approach may be of some importance in the case of large and/or deep vias.

One embodiment of the metallization process is illustrated schematically in FIG. 3a. First, a hole or track is printed (i.e., in the case of a hole, it is filled, and in the case of a track or trace, it is deposited on the PCB substrate or other layer) by laser jetting of a metal (e.g., Cu) (or otherwise) **302**, then the solvent is dried **304**, and the metal is sintered **306**. At this stage, the board can be taken to an imaging unit to verify that the desired hole or track thickness (e.g., in the case of a hole, the fill level) was reached **308**. If so, the board is passed to the next stage; otherwise, it is returned for additional printing/deposition and the process repeats until the desired amount of metal has been printed (i.e., a desired thickness reached). As shown in FIG. 3b, in other embodiments an additional ablation step to remove excess deposited material may be added before (or after) the sintering step.

At any layer, after metallization, a dielectric layer may be added to the board to reduce capacitance and avoid short circuits. There are several ways to add the dielectric layer, for example, by coating a liquid material and curing it or by hot pressing of a prepreg. Examples of such processes will now be explained.

Referring to FIGS. 4a and 4b, a dielectric layer **408**, which may be an epoxy and act as a passivation layer, may be deposited or coated over the metal (or other) layer and/or the PCB substrate with the aid of a roller or blade, see FIG. 4a, or by printing the dielectric layer from a donor film using the laser jetting system as shown in FIG. 4b. In the case of coating a liquid epoxy **402** using a roller **403** or blade, the epoxy may be a viscous liquid that includes a filler, such as silica balls. An amount of epoxy **402** is applied to one end of the board and the roller **403**, which may be made of or coated with an anti-sticking material such as polytetrafluoroethylene, ceramic, silicone, or other material, is placed at a desired height over the metal layer **214** and moved transversely over the board to spread the epoxy **402** into a layer **408** in a uniform manner at a desired thickness “h”. In some cases, the roller **403** may be fixed in position, and the board moved underneath it to cause the epoxy to spread. Where a blade applicator is used instead of a roller, the blade angle with respect to the board will affect the vertical force applied on the epoxy. If the angle is too small, the epoxy may not be squeezed into small apertures between portions of the metal layer. At the same time, if the blade pressure is too small, it may prevent the epoxy from being cleanly applied to the board and if it is too high, it may result in epoxy leakage outside the desired coverage area. Accordingly,

adjustment means for the blade angle should be provided and the blade angle adjusted according to the epoxy viscosity and other characteristics.

Alternatively, as shown in FIG. 4*b*, the epoxy 402 may be applied in a thin layer to a substrate or foil 404, and then deposited in small amounts 406 (e.g., droplets) onto the metal layer 214 and/or PCB substrate 104 by laser jetting using the same laser that produced beam 204 that was also used for jetting the metal layer. The epoxy 402 may be applied to the substrate 404, which is transparent or nearly so at the wavelengths of laser beam 204, by a roller system in which the substrate is passed between a pair of rollers or a single roller and a fixed surface separated by a well-defined gap so as to ensure the resulting coating of epoxy 402 is of uniform thickness. For example, such a coating system may include a syringe of the epoxy and an air or mechanical pump that drives the epoxy onto the substrate 404. The substrate 404 may then be moved towards the well-defined gap to create the uniform layer of epoxy 402 with a thickness that is defined by the gap. In some embodiments of the invention, the substrate 404 can translated bidirectionally in a controlled manner, while opening the gap between the coater rollers, creating the possibility for recoating the same area of the substrate 404 with the epoxy without contamination to the rollers and reducing or eliminating the amount of substrate 404 consumed during the coating process, thereby preventing waste.

Once coated, the substrate 404 with the layer of epoxy 402 thereon is positioned in the laser jetting system and dots 406 of the epoxy 402 are jetted onto the metal layer 214 and/or PCB substrate 104 using laser beam 204. In one example, the laser beam 204 is focused onto the interface between the layer of epoxy 402 and the substrate 404 causing local heating followed by a phase change and high local pressure which drives jetting of the epoxy onto the metal layer and/or PCB substrate. After printing the epoxy to the metal layer and/or PCB substrate, the substrate 404 can be returned for a second (or additional) coating of epoxy 402 by reversing the direction of a transport mechanism or, where substrate 404 is a continuous film, by moving substrate 404 through the coating system in a loop-like process.

In still further embodiments, the substrate 404 may be a screen or grid in which the epoxy 402 is introduced into holes of the screen by a coater, which may be a roller or blade, and the incident laser beam 204 used to displace the epoxy from the holes in the screen onto the metal layer and/or PCB substrate.

Referring to FIG. 4*c*, after printing or coating the epoxy layer 408, heat is applied to the layer by hot air, IR irradiation, and/or other heating method(s) and the epoxy layer is cured.

Yet another approach for passivation is to use a laminate to create a passivation layer. The use of laminate reduces height differences in a surface and creates a much more uniform height PCB surface. A laminate passivation layer can be formed in either of two different ways: printing the metal (e.g., Cu) onto the laminate and attaching the resulting structure to the surface (FIGS. 5*a-5c*) or attaching the laminate to the surface of the PCB and printing metal (e.g., Cu) into open vias (FIGS. 6*a-6c*).

In the first process, and referring to FIGS. 5*a-5c*, a laminate film 502 is attached to a liner 504 and a laser beam 506 is used to engrave/cut the laminate into a desired configuration, e.g., by creating through holes 508 and/or channels 510 in the laminate. The engraved areas are filled with metal (e.g., Cu) 202 using the laser beam 204 to deposit the metal from a film 208 coated on substrate 210, and only

then is the laminate 502 attached (e.g., by hot pressing) to the PCB substrate 104 (or a previously formed laminate layer) to create the layer of both metal and passivation. This approach can enhance the efficiency of the PCB build cycle significantly since the build of the layer and the attachment of the layer can be done as two independent stages, allowing serialization thereof.

Other advantages can be achieved from attachment of a laminate layer 602 (e.g., an epoxy laminate) to the PCB substrate 104 and subsequently engraving the laminate layer using a laser beam 606, as shown in FIG. 6*a*. This approach is more stable to thermal effect at the hot pressing stage when the laminate is attached to the PCB substrate. After engraving, the vias and holes are filled with metal in the fashion described above (FIG. 6*b*) and hot air 410 is applied to the surface of the board to dry the metal (FIG. 6*c*). Additional laser sintering can be added to enhance the conductivity of the metal particles.

In the final part of the PCB production the board is coated by a layer of solder mask. By acting as a protectant for underlying layers, the solder mask layer enables the soldering of electronic components to the board using high temperature without damaging the board's inner layers. As shown in FIG. 7*a*, one method for printing a solder mask 702 is by laser jetting. Similar to the manner in which an epoxy layer was printed, the solder mask material 704 may be applied in a thin layer to a substrate or foil 708, and then deposited in small amounts 710 (e.g., droplets) onto the metal layer 214 and/or PCB substrate 104 by laser jetting using a laser beam 706. Laser beam 706 may be produced by the same laser that was also used for jetting the metal layer and/or the epoxy layer. The solder mask material 704 may be applied to the substrate 708, which is transparent or nearly so at the wavelengths of laser beam 706, by a roller system in which the substrate is passed between a pair of rollers or a single roller and a fixed surface separated by a well-defined gap so as to ensure the resulting coating of solder mask material 704 is of uniform thickness. For example, such a coating system may include a syringe of the solder mask material and an air or mechanical pump that drives the solder mask material onto the substrate 708. The substrate 708 may then be moved towards the well-defined gap to create the uniform layer of solder mask material 704 with a thickness that is defined by the gap. In some embodiments of the invention, the substrate 708 can translated bidirectionally in a controlled manner, while opening the gap between the coater rollers, creating the possibility for recoating the same area of the substrate 708 with the solder mask material without contamination to the rollers and reducing or eliminating the amount of substrate 708 consumed during the coating process, thereby preventing waste.

Once coated, the substrate 708 with the layer of solder mask material 704 thereon is positioned in the laser jetting system and dots 710 of the solder mask material 704 are jetted onto the metal layer 214 and/or PCB substrate 104 using laser beam 706. In one example, the laser beam 706 is focused onto the interface between the layer of solder mask material 704 and the substrate 708 causing local heating followed by a phase change and high local pressure which drives jetting of the solder mask material onto the metal layer and/or PCB substrate. After printing the solder mask 702 to the metal layer and/or PCB substrate, the substrate 708 can be returned for a second (or additional) coating of solder mask material 704 by reversing the direction of a transport mechanism or, where substrate 708 is a continuous film, by moving substrate 708 through the coating system in a loop-like process.

Once the solder mask **702** has been printed, see FIG. *7b*, it is cured, as shown in FIG. *7c*, using UV light **714** and/or heat to create a mask layer that is open only in the soldering areas **712**.

It is often useful to have a label on top of the PCB board to help a technician understand the purpose of each component and to point the technician to areas on the PCB board. To that end, an additional label layer **802** may be added on top of the solder mask layer **702**. The label layer **802** can be printed by inkjet, screen printing, or other printing methods, and in one embodiment it is printed by the same laser jetting system used for printing the solder mask layer.

As shown in FIG. *8a*, the label material **804** may be applied in a thin layer to a substrate or foil **808**, and then deposited in small amounts **810** (e.g., droplets) onto the solder mask layer **702** by laser jetting using a laser beam **806**. Laser beam **806** may be produced by the same laser that was also used for jetting the metal layer, epoxy layer, and/or solder mask layer. The label material **804** may be applied to the substrate **808**, which is transparent or nearly so at the wavelengths of laser beam **806**, by a roller system in which the substrate is passed between a pair of rollers or a single roller and a fixed surface separated by a well-defined gap so as to ensure the resulting coating of label material **804** is of uniform thickness. For example, such a coating system may include a syringe of the label material and an air or mechanical pump that drives the label material onto the substrate **808**. The substrate **808** may then be moved towards the well-defined gap to create the uniform layer of label material **804** with a thickness that is defined by the gap. In some embodiments of the invention, the substrate **808** can be translated bidirectionally in a controlled manner, while opening the gap between the coater rollers, creating the possibility for recoating the same area of the substrate **808** with the label material without contamination to the rollers and reducing or eliminating the amount of substrate **808** consumed during the coating process, thereby preventing waste.

Once coated, the substrate **808** with the layer of label material **804** thereon is positioned in the laser jetting system and dots **810** of the label material **804** are jetted onto the solder mask layer **702** using laser beam **806**. In one example, the laser beam **806** is focused onto the interface between the layer of label material **804** and the substrate **808** causing local heating followed by a phase change and high local pressure which drives jetting of the label material onto the solder mask layer **702**. After printing the label layer **802**, the substrate **808** can be returned for a second (or additional) coating of label material **804** by reversing the direction of a transport mechanism or, where substrate **808** is a continuous film, by moving substrate **808** through the coating system in a loop-like process.

Once the label layer **802** has been printed, it is cured, as shown in FIG. *8b*, using UV light **814** and/or heat. The curing station may be the same one used for curing the solder mask layer.

One of the advantages of a PCB production system configured in accordance with embodiments of the invention is its ability to reduce the costs and production times associated with multi-layer PCB boards. Current production processes for boards that use more than one metal layer are complex. The present invention offers systems and methods which incorporate a wide range of applications that allow for straightforward and fast productions of boards with more than one metal layer. For example, while conventional PCB production processes do not accommodate the fabrications of a simple bridge between metal lines, such a bridge is

something that can be very easily done in systems configured in accordance with the invention.

FIG. *9a* illustrates an example of a PCB **900** that includes crossed metal (e.g., Cu) lines **902**, **904**. Using conventional PCB production processes, fashioning PCB **900**, if even possible, would entail several production stages, consuming significant production time. It may even require the use of a double-sided PCB. In a PCB production system employing the methods of the present invention, however, production complexity and time are significantly reduced. For example, using one or more of the above-described techniques, epoxy patches **910**, **912** may be printed using laser jetting, allowing the printing of metal lines **902**, **904** on a single side of PCB **900** and, optionally, using the same laser jetting apparatus as is used to print the epoxy patches. FIG. *9b* provides a close-up three-dimensional view of one of the wire crossings on PCB **900** illustrated in FIG. *9a*. After printing metal line **902**, the epoxy patch **910** can be printed so that subsequent metal line **904** passes over line **902** on the same side of PCB **900**, without creating a short circuit. This simple example illustrates how complex double-sided boards of the past can be fashioned in a relatively straightforward manner using the laser jetting techniques for different materials as discussed above. Of course, the present systems and methods may also be used to fashion double-sided PCBs, with or without bridging structures such as epoxy patch **910**, thereby facilitating the production of single and double-sided boards, with bridged and non-bridged areas.

Turning now to FIGS. *10a* and *10b*, examples of PCB processing systems **1000a** and **1000b** configured in accordance with embodiments of the present invention are illustrated. FIG. *10a* illustrates system **1000a**, composed of individual sub-units, while FIG. *10b* illustrates system **1000b**, in which the sub-systems are arranged into various modules. In these examples, PCB processing systems **1000a** and **1000b** include an imaging sub-system **1003** and a laser sub-system **1004**, which together may be organized into a printing unit **1002**. A UV light sub-system **1008** may be included in a UV curing unit **1006**, although this is an optional component. A heating sub-system **1012** may be a component of a heating unit **1010**. Further, a drilling unit **1014**, which includes the CNC drilling head **102** described above and a three-dimensional translation sub-system **1016** for maneuvering the drilling head may be provided as a module, or the components provided separately. Optionally included in the drilling unit **1014** or elsewhere may be a PCB flipping sub-system **1018** which is configured to flip the PCB during processing so as to allow for drilling, printing, etc. on both sides of the PCB. Where laminates are used, an optional hot press sub-system **1020** is available. Associated with the printing unit **1002** are various materials **1022** for the LAD procedures discussed above. These include the metal used for conductive traces (e.g., Cu), solder mask material, epoxy(ies), etc. as well as the donor substrate(s) on which these materials are coated for deposition onto the PCB. Although not shown in the diagrams, while the printing unit **1002** can include only a laser sub-system that is for all the laser deposition processes, it may also include an inkjet head or screen printer for printing labels or solder mask materials.

Although not discussed in detail above, imaging sub-system **1003** may be employed in connection with any or all of the above-described etching and deposition procedures. For example, the imaging sub-system may include one or more two-dimensional and/or three-dimensional imaging units (e.g., cameras, scanning laser arrangements, etc.) that image the PCB or portions thereof at various stages during the production process. Vias and holes or features may be

imaged so as to ensure they are free from debris and regular in shape. Deposited layers may be imaged so as to ensure they are uniform in coverage and/or accurately positioned. This may be especially important where layers are printed through successive jetting of small droplets of material. The imaging may also be used to ensure accurate registration of the PCB substrate **102** on a holder **120**. Imaging in this fashion can allow for in-line repair of a process step, such as additional or re-coating of a layer, or rejection of an in-process PCB when necessary. Stage **1030** which can translate in two dimensions and, where necessary, raise and lower PCB **102**, facilitates movement of the PCB between the various units of system **1000** and the sub-systems within those units during processing.

As mentioned, the UV light sub-system **1008**, whether modularized or not, is optional. As all of the deposited layers can be heat-cured, the use of UV curing is not mandatory, hence, the need for the UV light sub-system is only in cases where UV curing is preferred. When modularized, the UV light sub-system **1008** can be included in the overall system or removed therefrom as desired.

The heating sub-system **1012** is used for curing heat sensitive materials and/or for drying solvent based materials. It can be a part of an overall system **1000a**, but it preferably is modularized **1010** so that it can be easily replaced, if necessary, in a system **1000b**.

The drilling unit **1014** is used to create holes and vias in the laminates as discussed above. As previously noted, the drilling unit **1014** can be a CNC head **102**, or a laser based cutting and engraving unit or a combination of both. While the CNC head and/or laser cutting tool can be incorporated in a larger system such as **1000a**, it is preferable that a modular drilling unit **1014** be used so as to compartmentalize the debris generated during the drilling process. After drilling a cleaning procedure is applied to the PCB board either by adding a protective film prior to drilling, by cleaning the surface with air pressure and suction, or by another cleaning process.

In cases where a solid laminate is used as a base for the PCB, the additional hot press sub-system **1020** is used for attachment of the laminate to the PCB substrate **104**.

FIGS. **11a-11g** illustrate aspects of a production cycle of a single-sided, multilayer PCB, and FIGS. **12a-12d** illustrate aspects of a production cycle of a double-sided, multilayer PCB, each from substrate to a final board in accordance with embodiments of the present invention. Beginning with the single-sided, multilayer PCB **1100** illustrated in FIG. **11a**, the PCB substrate **1102** may be an FR4 board or a flexible board. A first copper (or other) cladding layer **1104** is formed on top of PCB substrate **1102**, and is covered by an epoxy layer **1106a**. One or more through holes **1114** may be provided in the epoxy layer **1106a** to electrically connect copper cladding layer **1104** to a second copper (or other metal) structured layer **1108** disposed on the upper side of epoxy layer **1106a**. The through holes may be formed by a drilling process and copper or other metal connectors deposited therein as described above. The upper copper structured layer **1108** is also covered by an epoxy layer **1106b**. This structure may be repeated for as many copper (or other metal) layers as are needed, with through holes **1116** having copper or other metal connectors provided as required. Ultimately, the top-most copper layer is covered by a solder mask layer **1110** and gold, silver, or other metal connectors **1112** may be added to allow electrical connections to the various copper lines **1104**, **1108** via the metalized through holes **1116**, **1114**. Gold, silver, or other metal connectors **1112** enable the soldering of electrical components to the

PCB board. Examples of processes for forming the single-sided, multilayer PCB **1100** and components thereof will be discussed with reference to FIGS. **11b-11g**.

As shown in FIG. **11b**, fabrication of the single-sided, multilayer PCB begins with obtaining a PCB substrate (FR4 or other) **1102** having a copper (or other metal) cladding layer **1104** disposed on one side thereof. It is possible to purchase PCB substrates with a copper cladding layer and, in one embodiment of the invention, these may be used as a starting material and the copper cladding layer etched by laser to remove excess material, leaving the desired Cu line formed on the PCB substrate. Alternatively, as shown in the illustration, a bare substrate (FR4 or other) **1120** may be provided to a laser jetting station such as one described above, and copper (or other metal) lines created on one side thereof through laser jetting of a copper layer **1122** coated on a donor substrate **1124** using a laser beam **1126**. Once any solvent has dried, the copper layer **1128** jetted onto the surface of the PCB substrate **1120** is sintered and excess copper removed through ablation. The sintering and ablation processes may be performed using the same laser **1126**, albeit at different energy than is used for the jetting process. Alternatively, a different laser may be used for sintering and ablation.

After the first copper layer has been formed, a second layer is added to form connectors to additional layers. FIG. **11c** illustrates one example for printing copper connectors **1134** on top of a first copper layer **1132** on a PCB substrate **1130**. In this example, printing is by laser jetting of copper droplets from a copper film or coating **1138** carried on a donor substrate **1140**, in the fashion described above. A sufficient number of droplets are jetted for each connector **1134** using a laser **1136**. Thereafter, the connectors **1134** are sintered and, if necessary, ablated to form precise shapes having desired dimensions. As before, the sintering and ablation may make use of the same laser **1136** used for printing the connectors, at different energies, if needed. Following sintering and ablation of the copper connectors, an epoxy layer **1146** is added by laser jetting of an epoxy coating **1142** carried on a donor substrate **1144**. The same laser **1136** may be used to print the epoxy layer **1146** and subsequently, the epoxy layer **1146** is heated **1148** in order to cure the layer.

In an alternative process, illustrated in FIG. **11d**, the epoxy layer **1146** is printed by laser jetting onto the first copper layer **1132**, cured, and then holes/vias **1148** are ablated in the epoxy layer in locations at which connectors will ultimately be formed. Thereafter, the copper connectors **1134** are printed by laser jetting in the holes/vias **1148** that were created in the epoxy layer. The structure is completed with additional drying and sintering. The processes presented in FIGS. **11c** and/or **11d** may be repeated multiple times to create a multilayered structure **1150**, illustrated in FIG. **11e**.

Once a desired number of metal (e.g., copper) and epoxy layers have been formed, electrical connectors and a solder mask layer may be printed. FIG. **11f** illustrates one example of such procedures. First, gold connectors **1152** are printed by laser jetting on top of the copper connectors **1134** to create a conductive connection. The laser jetting process uses a laser beam **1158** to jet gold droplets from a gold film or paste **1154** coated on a donor film or substrate **1156** as described above. Then, the gold connectors **1152** are dried and sintered, and ablated if needed, and a solder mask layer **1160** is jet printed on the surface epoxy layer and cured either heat and/or UV light **1166**. The laser used for the sintering, ablation (if needed), and printing of the solder

mask layer may be the same as is used for printing the gold connectors and/or used for printing the copper connectors, with energies adjusted accordingly.

As shown in FIG. 11g, the same result may be achieved by first jet printing the solder mask layer 1160, curing it by heating or UV irradiation 1166, and then ablating the solder mask layer 1160 to create holes 1168 for placing the gold connectors 1152. The gold connectors 1152 are printed and sintered as described above.

The process for forming a double-sided, multilayer PCB is similar to that as described above for creating a single-sided, multilayer PCB, except that some accommodations are needed due to the need for drilling from both sides. Referring to FIG. 12a, a double-sided, multilayer PCB 1200 includes a PCB substrate 1202 on each side of which are one or more metal (e.g., copper) layers (traces) 1202, each separated by a layer of epoxy 1204. On each side, the final epoxy layers are covered by a solder mask layer 1206 and gold (or other metal) connectors 1208 are added at positions at which electrical components will be connected. The gold connectors 1208 are electrically connected to lower copper layers by copper-(or other metal-) filled vias and holes 1210 in the epoxy layers.

There are different available alternatives for providing the holes on each side of the PCB substrate 1202. Referring to FIG. 12b, in one embodiment, one or more through holes 1220 may be drilled in PCB substrate 1202, with the drilling tool passing through the entire thickness of the PCB substrate. In such a case, it is preferable to subsequently provide a peelable substrate 1222 beneath the PCB substrate 1202 after the drilling is complete in order to accommodate the jet printing of the copper (or other metal) filler into the hole, without contaminating the sample holder. Also, a sample holder 1224 that secures the PCB substrate from its edges should be used. Using a laser 1228, copper (or other metal) is jet printed from a film or paste 1230 on a donor substrate 1232 in the fashion described above so as to fill the hole 1220. Subsequently, the copper fill 1226 is sintered (e.g., using the same laser 1228) in place in the hole. Thereafter, the peelable substrate 1222 and sample holder 1224 may be removed.

Referring to FIGS. 12c and 12d, another option is to drill a hole 1240 in one side of the PCB substrate 1202, taking care that the hole 1240 does not pass all the way through the entire thickness of the PCB substrate. As before, the hole 1240 is filled with copper 1226 from a Cu film 1230 carried on a donor substrate 1232 by jet printing, and the Cu fill is subsequently dried, sintered, and ablated (if necessary). Thereafter, referring to FIG. 12d, the PCB substrate 1202 is flipped, and the laser 1228 is used to complete the through hole by removing the remaining portion 1242 of the PCB substrate opposite the Cu fill 1226. This leave a small hole 1244 to be filled with copper by jet printing, as above, and the printed copper 1246 is again dried and sintered as discussed above.

It is also possible to add a pick and place machine to the system so as to provide for full assembly of the PCB with electronic components. FIGS. 13a-13i illustrate several examples of the inclusion of embedded and other electronic components facilitated through the use of such a machine in accordance with embodiments of the present invention. In these examples, a single-sided, multi-layer PCB is discussed. This is primarily for ease of illustration; however, it should be appreciated that the discussion applies equally to double-sided, multilayer PCBs fashioned in accordance with the present invention. In the case of double-sided PCBs, the board may be flipped when working on a reverse side, or

pick and place apparatus may be provided on both sides of a PCB supported at its edges so as to allow component situation and mounting on both sides simultaneously or nearly so.

FIG. 13a illustrates a PCB assembly 1300, which includes a PCB substrate 1302 made of, for example, FR4 or a flexible board, on which is disposed a plurality of metal (e.g., copper) layers 1304 separated by epoxy layers 1306. The top-most epoxy layer is covered by a solder mask layer 1308 in which are located a plurality of gold (or other metal) connectors 1310, which are electrically coupled to metal (e.g., copper) connectors 1312. PCB assembly 1300 may be fabricated using the above-described PCB production systems and methods.

Also included in PCB assembly 1300 are a plurality of electronic components 1316a-1316c. Components 1316a-1316c may be integrated circuits (ICs) or other electronic components and are assembled into PCB assembly 1300 using a pick and place machine or similar system. In particular, components 1316a-1316c are situated so that leads associated therewith make electrical connections to connectors 1310, 1312, e.g., at solder pads 1314 or elsewhere. Solder pads 1314 may be deposited by LAD in a manner similar to the deposition of copper traces, solder masks, and/or connectors, as described above.

As shown, electronic components 1316a-1316c may be added to the PCB assembly at the top or sides of the structure and/or embedded within it. When added from the or sides, the electronic components may be electrically connected to the metal layers by connectors, as discussed above, or directly, as is the case with electronic component 1316c in the illustration. It is also possible to encapsulate an electronic component 1316b within the PCB assembly 1300. FIGS. 13b-13f further illustrate aspects of these various procedures for electrically connecting the electronic components.

In FIG. 13b, a PCB substrate 1302 has been printed with a metal (e.g., copper) layer 1304 and one or more epoxy layers 1306 in the manner discussed above. Metal (e.g., copper) connectors 1312 have been printed in through holes in the epoxy layer(s) according to the processes described above. The connectors 1312 have been positioned to accommodate the placement of an electronic component. In one embodiment, the structure shown in FIG. 13b may be fashioned by selectively depositing epoxy by jetting so as to leave an open area 1320 for placement of the electronic component. Alternatively, the open area 1320 may be created through laser engraving (e.g., etching or ablation) of a layer of epoxy.

FIG. 13c shows a next step in preparing the structure to receive an electronic component. A solder mask layer 1308 has been deposited by LAD in the open area 1320 on top of the epoxy layer, and gold connectors 1310 have been fashioned therein so as to provide electrical coupling to the copper connectors 1312 below. Then, as shown in FIG. 13d, solder paste 1314 is jet printed onto the gold connectors 1310. The solder paste is printed by laser jetting using a laser 1330 and a donor substrate 1332 with solder 1334 coated thereon in the manner discussed above. An electronic component 1316d is then placed in the open area 1320 with its leads electrically connected to the solder paste areas 1314, for example using a conventional pick and place machine, as shown in FIG. 13e. Placement of the electronic component in this fashion may be observed using one or more imaging stations so as to ensure correct alignment of the leads with the solder paste locations.

Through the printing of additional epoxy layers **1306'**, copper connectors **1312'**, gold connectors **1310'**, solder paste areas **1314'**, and (optionally) solder mask areas), additional electrical components **1316e** may be added to the PCB assembly, as shown in FIG. **13f**. As shown, electrical connections may be made to multiple sides of an electronic component **1316d**, and in some cases additional metal (e.g., CU) layers, labels, etc. may be added as described above. During fabrication of this PCB assembly, support structures **1330** made of resin or other material may be added to provide temporary or permanent support for layers overlying open areas **1320**. The support structures may be added by LAD in the manner discussed above for the printing of other layers.

The assembly of a PCB structure is not limited to the placement of electronic components. Other structural components may be included in such a structure. FIG. **13g** illustrates an example in which a heat dissipation element **1340** has been added by adhering the heat dissipation element (e.g., a metal plate or other thermal conductor) to electronic component **1316d** by a layer of thermal conductive glue **1342**. The thermal conductive glue may be printed by LAD in a fashion similar to that described above for the printing of metal layer or connectors, or solder masks or epoxies. The heat dissipation element may be placed by a pick and place machine or, in some cases, may be a printed structure formed by LAD.

Nor is the assembly of a PCB structure limited to placement of electronic components in layer arrangement with other elements of the PCB structure. FIG. **13h** illustrates placement of an electronic component **1316f** such that its electrical leads are orthogonal to the plane of the PCB substrate **1302** and its overlying metal, epoxy, and other layers, and which are electrically connected via metal (e.g., Cu) connectors **1352**, gold connectors **1354**, and solder paste areas **1356** that are printed so as to accommodate this orientation. As shown, a temporary or permanent support structure **1330** may be printed so as to be used as a support for the electronic component when placed in this fashion. The printing of the various epoxy layers **1350**, connectors **1352**, **1354**, and solder paste **1356** may be done in the manners discussed above using LAD. It may be necessary to maintain a portion of an epoxy layer in place while printing the solder paste, for example, so as not to contaminate structures and/or assemblies below. Although not shown, a solder mask layer may be printed, if needed.

And still another example of a PCB assembly accommodating embedded electronic components is illustrated in FIG. **13i**. In this case, a coil **1360** is embedded in an epoxy layer **1362**. The coil may be an existing component that is placed by a pick and place machine and electrically connected to another electronic component **1316d** via metal (e.g., Cu) connectors **1310"**, gold connectors **1312"**, and solder joints **1314"**, which themselves are printed in the fashions discussed above, or the coil **1360** itself may be printed by LAD, for example as described in the present assignee's U.S. patent application Ser. No. 16/946,638, filed 30 Jun. 2020. For example, the coil may be printed by jetting metal from a coating or foil on a donor substrate, similar to the method discussed above for printing metal layers, with the jetted material printed in successive layers so as to partially overlap a most-recently printed element and thereby define a helical pattern of coil elements that collectively make up the coil **1360**. The coil elements may be printed within the supporting epoxy **1362** and/or about an inner core (not shown), which act (singularly or collectively) as scaffolds to keep the coil under construction intact as the

coil elements fuse with one another. Alternatively, the coil **1360** may be printed by LAD as a plurality of partially complete rounds, each in a respective layer, with pillars interconnecting successive ones of the partially complete rounds in different ones of the respective layers. The pillars may be vertical, or near-vertical and positions of the pillars between successive ones of the plurality of partially complete rounds may be staggered across the circumference of the partially complete rounds. Following printing of one of the plurality of partially complete rounds in a respective layer, for a number of successive layers of metal corresponding to a desired pillar height, only a connecting pillar is printed. Epoxy scaffolding elements may be printed as part of each respective layer of material concurrently with printing the plurality of partially complete rounds. Embedding coils in this fashion may provide for embedded radio frequency identification (RFID) tags within the PCB assembly.

FIGS. **14a-14d** illustrate aspects of a production cycle of a PCB in which laser assisted deposition or another laser-based printing process is used for some aspects of PCB fabrication. In FIG. **14a**, metal layers (e.g., Cu) **1404**, **1406** are printed on both sides of a PCB substrate **1402** using LAD or another laser-based printing process, as described above. In FIG. **14b**, the double-sided PCB from FIG. **14a** is stacked with a similarly fabricated double-sided PCB consisting of metal layers (e.g., Cu) **1412**, **1414** printed on both sides of a PCB substrate **1410** with a prepreg layer **1416** between them. The prepreg layer **1416** may be a conventional resin-based prepreg (and may be made of the same material as the PCB core substrate), and the stacking process may be performed by conventional means used in the PCB fabrication industry. The result is an assembly **1420**.

As shown in FIG. **14c**, the stacking process may be repeated with additional double-sided PCBs **1430**, **1440**, etc., each time including a prepreg layer **1450** between the new double-sided PCB and the previous assembly. When the assembly includes two or more double-sided PCBs with interdigitated prepregs, as shown in the illustration, it is subjected to a lamination process using a conventional hot pressing process and a photosensitive dry resist. The result is a multi-board stack **1460**. Then, as shown in FIG. **14d**, solder mask layers **1470** may be printed on each side of the multi-board stack using LAD or another laser-based printing process, as described above. Although not shown in detail, gold, silver, or other pads, and solder as well as labels may also be printed on each side of the multi-board stack using LAD or another laser-based printing process, as described above. Drilling of the PCBs and/or copper plating following drilling may be performed using conventional processes for such activities.

Although not illustrated in detail, it should be appreciated that the various components of the printing systems described herein operate under the control of one or more controllers, which, preferably, are processor-based controllers that operate under the instruction of machine-executable instructions stored on tangible machine-readable media. Such controllers may include a microprocessor and memory communicatively coupled to one another by a bus or other communication mechanism for communicating information. The memory may include a program store memory, such as a read only memory (ROM) or other static storage device, as well as a dynamic memory, such as a random-access memory (RAM) or other dynamic storage device, and each may be coupled to the bus for providing and storing information and instructions to be executed by the microprocessor. The dynamic memory also may be used for storing temporary variables or other intermediate information dur-

19

ing execution of instructions by the microprocessor. Alternatively, or in addition, a storage device, such as a solid state memory, magnetic disk, or optical disk may be provided and coupled to the bus for storing information and instructions. The controller may also include a display, for displaying information to a user, as well as various input devices, including an alphanumeric keyboard and a cursor control device such as a mouse and/or trackpad, as part of a user interface for the printing system. Further, one or more communication interfaces may be included to provide two-way data communication to and from the printing system. For example, network interfaces that include wired and/or wireless modems may be used to provide such communications.

What is claimed is:

1. A system for fabricating a printed circuit board (PCB) assembly, the system comprising:

a substrate holder configured to hold a PCB substrate and translatable between a plurality of processing stations, said processing stations including a printing station;

the printing station configured for laser-assisted deposition (LAD) of one or more materials by jetting respective ones of said materials individually from respective donor substrates on which said respective ones of said materials are coated or otherwise disposed;

a curing station configured to cure by heating, infrared (IR) irradiation, or ultraviolet (UV) irradiation, deposited ones of said materials on said PCB substrate; and

20

a drilling station configured to drill or engrave through holes and/or vias in said PCB substrate and/or layers of ones of said materials disposed thereon,

wherein the donor substrate includes a first donor substrate and a second donor substrate, and

wherein the printing station is configured to jet metal droplets from the first donor substrate onto the PCB substrate and/or into one or more holes therein to form a layer of metal on the PCB substrate using a laser of the printing station and to jet an epoxy from the second donor substrate onto the layer of metal to form a layer of the epoxy on the layer of metal using the laser of the printing station.

2. The system for fabricating a PCB assembly as in claim 1, wherein said printing station is further configured for laser sintering and/or laser ablation of said respective ones of said materials printed on said PCB substrate and/or additional layers of said PCB assembly.

3. The system for fabricating a PCB assembly as in claim 1, wherein said materials include copper and gold pastes, the epoxy, and solder mask material.

4. The system for fabricating a PCB assembly as in claim 3, further comprising a unit configured to flip the PCB substrate to allow access to both sides of the PCB substrate by the printing station, curing station, and/or drilling station.

* * * * *